GSK983M Milling CNC System

User Manual

(Volume II: Operations)

广州数控设备有限公司

GSK CNC EQUIPMENT CO., LTD.

This user manual describes all items concerning the operation of the system in detail as much as possible. However, it is impractical to give particular descriptions of all unnecessary and/or unavailable works of the system due to the length limit of the manual, specific operations of the product and other causes. Therefore, the operations not specified herein may be considered impractical or unavailable.

This user manual is the property of GSK CNC Equipment Co., Ltd. All rights are reserved. It is against the law for any organization or individual to publish or reprint this manual without the express written permission of GSK and the latter reserves the right to ascertain their legal liability.

Company Profile

GSK, GSK CNC Equipment Co,. Ltd, is the largest CNC system production and marketing enterprise in China at present. It is the Numerical Control industrial base of South China, and the undertaking enterprise of the national 863 main project Industrialization Support Technology for Medium Numerical Control System. It is also one of the 20 basic equipment manufacture enterprises in Guangdong province. It has been taking up the research and development, design and the manufacture of machine CNC system (CNC device, drive unit and servo motor) in recent 10 years. Now it has developed into a large high-tech enterprise integrated with technology, education, industry and trade by enhancing the popularization and trade of CNC machine tools. There are more than 1400 staffs in this company that involves 4 doctors, more than 50 graduate students and 500 engineers; more than 50 among these staffs are qualified with senior engineer post titles. The high performance-cost ratio products of GSK are popularized in China and Southeast Asia. And the market occupation of GSK's product dominates the first and the turnout and sale ranks the top for successive 7 years in domestic market for the same product from the year 2000 to 2006, which makes GSK the largest CNC manufacture base throughout China.

Field technical support services

Field support services are available when you encounter a problem insolvable through telephone. GSK CNC Equipment Co. Ltd will designate a technical support engineer to your place to solve technical problems for you.

Chinese version of all technical documents in Chinese and English languages is regarded as final.

Foreword

Dear user,

We are really grateful for your patronage and purchase of GSK983M milling CNC system made by GSK CNC Equipment Co., Ltd.

This manual consists of two volumes. Volume I mainly describes the specifications and programming of the system while Volume II operations, all codes, parameters, I/O interfaces and other appendices.

This system can only be operated by authorized and qualified personnel as improper operations may cause accidents. Please carefully read this user manual before usage.

All specifications and designs herein are subject to change without further notice.

We are full of heartfelt gratitude to you for supporting us in the use of GSK's products.

CONTENTS

4. OPERATIONS.....1

4.1 Power On/off 1
4.1.1 Power on 1
4.1.2 Power off 1

4.2 Key Switch.....1

4.3 Operations Involving with the Operation Panel 1
4.3.1 Operation panel.....1
4.3.2 Emergency stop (red).....2
4.3.3 Selection mode.....3
4.3.4 Operations involving with manual operation.....3
4.3.4.1 Jog feed.....3
4.3.4.2 MPG feed.....4
4.3.4.3 Manual absolute 5
4.3.5 Manually returning to reference point (reference position) 9
4.3.6 Operations on automatic running 10
4.3.6.1 Start of automatic operation.....10
4.3.6.2 Suspension of automatic operation.....10
4.3.6.3 Single block 10
4.3.6.4 Restart after feed hold or stop 11
4.3.6.5 The manual operations in automatic run.....11
4.3.6.6 The MDI operations in automatic run.....11
4.3.6.7 Skipping over optional blocks 12
4.3.6.8 Feedrate override 12
4.3.6.9 Dry run.....12
4.3.6.10 Machine lock.....12
4.3.6.11 Display lock.....12
4.3.6.12 Mirror image.....12
4.3.6.13 Rapid traverse override 13
4.3.6.14 External workpiece number search function 13
4.3.7 JOG feed at any angle 14
4.3.8 Manual insertion.....15
4.3.8.1 Inserting operations by MPG(manual pulse generator/handwheel) 15
4.3.8.2 Manual inserting movement by MPG.....16

4.4 The Display and Operations of the MDI/LCD Panel with LCD Characters 17
4.4.1 Status display 18
4.4.2 Key input 18
4.4.3 Display of program numbers and sequence numbers 19
4.4.4 Alarm display (function button ALARM).....20
4.4.5 Operator information 21
4.4.6 Display of actual position and reset (function key POSITION) 21
4.4.7 Indication of instruction value (function button COMMAND) 22
4.4.8 Setting (function button SETTING) 24
4.4.8.1 Display and setting of input, output, etc 24
4.4.8.2 Display and setting of user macro variables 26
4.4.9 Operating through MDI (function key COMMAND) 27
4.4.10 Start of MDI motion.....28
4.4.11 Reset 28
4.4.12 Tool position offset.....29
4.4.13 Setting and display of workpiece origin offset (Optional).....30
4.4.14 Measurement of tool length 31
4.4.15 Program display (function button PROGRAM).....31
4.4.16 Program number search (function key PROGRAM) 33
4.4.17 Inputting a program with keys.....33

4.4.18 Deletion of a program....35
4.4.19 Deletion of all programs....35
4.4.20 Sequence number search....35
4.4.21 Restart of a program....36
4.4.22 Program number comparison stop function....39
4.4.23 Display of parameters (function button: PARAMETER)....40
4.4.24 Program edit....40
4.4.24.1 Word scanning....41
4.4.24.2 Word search....42
4.4.24.3 Address search....42
4.4.24.4 Methods for returning to the beginning of a program....42
4.4.24.5 Word insertion (active when the program protection lock is disabled)....43
4.4.24.6 Word modification (active when the program protection lock is disabled)....44
4.4.24.7 Insertion and modification of several words, blocks and strings....44
4.4.24.8 Word deletion (active when the program protection lock is disabled)....45
4.4.24.9 The deletion of the part before EOB....45
4.4.24.10 Deletion of blocks (active as program protection lock is disabled)....45
4.4.24.11 Storage sorting....46
4.4.24.12 The indication of all stored program numbers....46
4.4.24.13 Edit of user macro (active as the program protection lock is disabled)....46
4.4.25 Indication of run time....48
4.4.26 Menu switching function....48
4.4.27 Operations of LCD soft function keys....50
4.4.27.1 General....50
4.4.27.2 Display....50
4.4.27.3 Direct entry of measured workpiece origin offset....57
4.5 Position Indication through Position Display Unit (Available upon Customer's Request)....58
Appendix 1 Codes for Programming....59
Appendix 2 G Codes List....62
Appendix 3 Ranges of Instruction Values....64
Appendix 4 Calculating Chart....65
Appendix 5 Parameters....69
Appendix 6 Alarms List....108
Appendix 7 List of the States during Switching On, Reset and Clearance....117
Appendix 8 Memory Type Pitch Error Compensation....119
Appendix 9 Operations List....125
Appendix 10 Lock of Program Key....127
Appendix 11 The Interrupt Function of User Macro....129

4. OPERATIONS

4.1 Power On/off

4.1.1 Power on

1) Make sure all parts of the machine are properly wired and secured.
2) Switch on the machine by following its manual.
3) Pictures appear on the LCD several seconds after switching on the machine.

4.1.2 Power off

1) The indicator of the cycle start button on the operation panel of the machine goes out.
2) All moving parts of the machine stop.
3) Make sure the above operations are performed well and then press down and hold the POWER OFF button for 1 or 2 seconds.
4) Disconnect the power supply of the machine by following its manual.

   Note: Never use the keys on the MDI keypad to power on/off the machine.

4.2 Key Switch

A key switch for program protection may be set with the operation panel of the machine. The key switch offers two modes of protection:

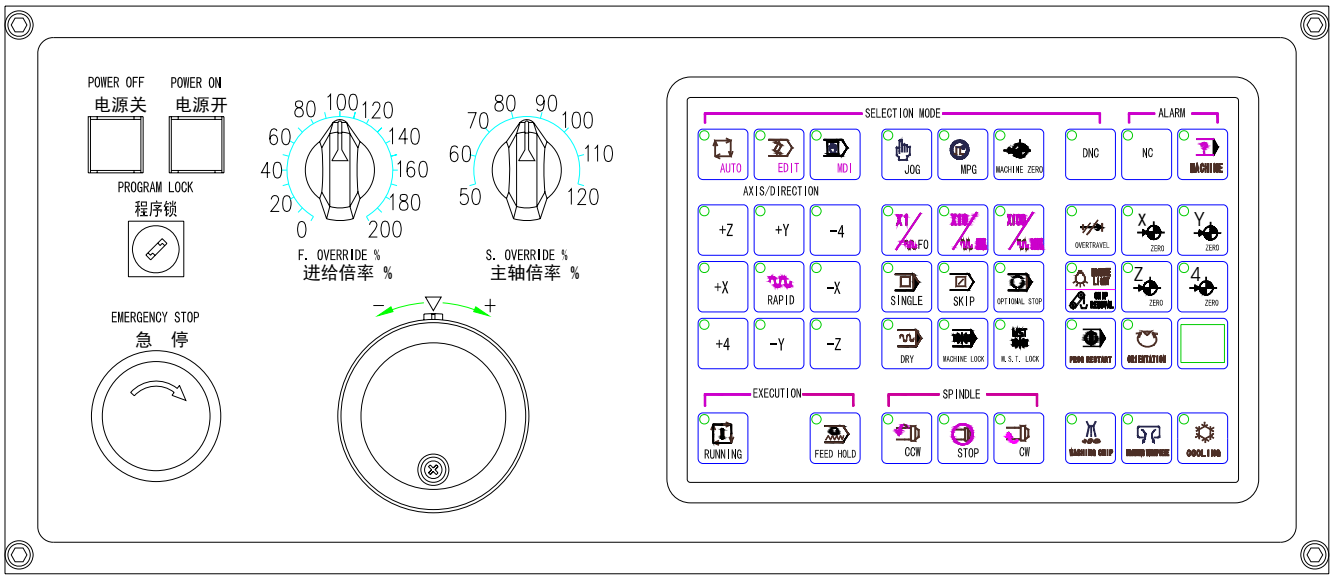
1) Relevant operations cannot be performed unless the key switch is actuated. However, the concerned data is still displayed on the LCD.
2) Operations can or cannot be performed without actuating the key switch. It is possible to switch between the two modes by parameter.

   The Section 4.4 herein will describe in detail which functions are under the protection of 1) or 2) mode.

4.3 Operations Involving with the Operation Panel

4.3.1 Operation panel

The functions of the operation panel and the layout of switches on it vary depending on different machine types. The following is a typical operation panel. Refer to the relevant parts of the manual supplied with the machine for details. This chapter only describes the operation panel of 3-axis control system (The operation panels of 4-axis and 5-axis control systems are primarily similar to that of a 3-axis control system).



4.3.2 Emergency stop (red)

In an emergency, press the EMERGENCY STOP button to stop the movements of all the axes of the machine. At the same time, the button is locked in the stop position.

The release mode of the button varies with different manufacturers. In general, it is released by pushing down and clockwise turning the button.

Note 1: The power supply of the motor is switched off when the button is pressed.

Note 2: The control unit is in reset state.

Note 3: Make sure to eliminate all troubles before releasing the button.

Note 4: Return to the reference point by through manual operations or G28 instruction.

4.3.3 Selection mode

Modes	Functions
EDIT	Perform the following operations: (1) Saving programs in storage; (2) Modifying, inserting and deleting programs; (3) Outputting the programs in storage and editing other programs
AUTO (MEMORY)	(1) Executing the programs saved in storage; (2) Search the sequence numbers of the programs in storage
MDI	(1) Manual data entry may be performed through MDI and the operation panel of the machine.
JOG	(1) It is possible to perform Jog feed.
MPG(HANDLE)	(1) It is possible to perform manual feed.
MACHINE ZERO (HOME)	Return to the machine zero.

4.3.4 Operations involving with manual operation

Except the automatic operations that can be performed with programs, it is possible to conduct the following manual operations with switches.

4.3.4.1 Jog feed

Jog feed enables the machine to move.

1) Set the selection mode switch to JOG position.

2) Select a motion axis so that the machine travels in the selected direction.

Note 1 2 axes may be concurrently controlled by manual operation.

Note 2 After power on, the selected axis of the machine will not immediately move even the SELECTION MODE switch is set to JOG position. Now it is necessary to reselect an axis.

3) Selecting Jog feedrate

Position of switch	Feedrate			
	Lead screw for feed in metric system		Lead screw for feed in Inch system	
	mm/min	inch/min	inch/min	mm/min
0	0	0	0	0
1	1.0	0.04	0.02	0.508
2	1.4	0.055	0.208	0.711
3	2.0	0.079	0.04	1.02
4	2.7	0.106	0.054	1.37
5	3.7	0.146	0.074	1.88
6	5.2	0.205	0.104	2.64
7	7.2	0.283	0.144	3.66
8	10	0.394	0.2	5.08
9	14	0.551	0.28	7.11
10	20	0.787	0.40	10.2
11	27	1.06	0.54	13.7

12	37	1.46	0.74	18.8
13	52	2.05	1.04	26.4
14	72	2.83	1.44	36.6
15	100	3.94	2.00	50.8
16	140	5.51	2.80	71.1
17	200	7.87	4.00	102
18	270	10.6	5.40	137
19	370	14.6	7.40	188
20	520	20.5	10.40	264
21	720	28.3	14.40	366
22	1000	39.4	20.00	508
23	1400	55.1	28.00	711
24	2000	78.7	40.00	1016

Note 1: The numerical values listed in the above table vary with different machines.

Note 2: The allowable deviation of the feedrates listed in the above table is approximately ±3%.

4) Rapid traverse

An axis rapidly traverses in the selected direction when the button is pressed.

Note 1: The feedrate, time constant and acceleration/deceleration mode for manual rapid traverse are the same as the rapid traverse under G00 program instruction.

Note 2: When the machine has a memory type travel limit selecting function, it shall be provided with an axis with the function of returning to the reference point. When the RAPID button is pressed after power on or emergency stop, its feedrate will not change into rapid feed but maintain at Jog feedrate provided that the function of returning to the reference point is not executed.

This is because memory type travel limit dose not function before the manual return to the reference point, thereby preventing the machine from quickly reaching the end of run.

4.3.4.2 MPG feed

Make accurate adjustment for the feed of the machine with a manual impulse generator as follows.

(1) Set the SELECTION MODE switch to MPG position.

(2) Select a motion axis.

(3) Turn the handwheel of manual impulse generator.

Clockwise……..+ direction

Counterclockwise…… - direction

(The rotating direction depends on the settings of manufacturers)

(4) Amount of travel: Some of the operation panels are provided with the following selector switches: ×10 means multiplying amount of travel by 10 while ×100 by 100.

Input system	The amount of travel each step		
	×1	×10	×100
Input in metric system	0.001mm	0.01mm	0.1mm
Input in inch system	0.0001inch	0.001inch	0.01inch

Note 1: If the handwheel rotates at a speed over 5 turns per second, the amount of the rotation of the

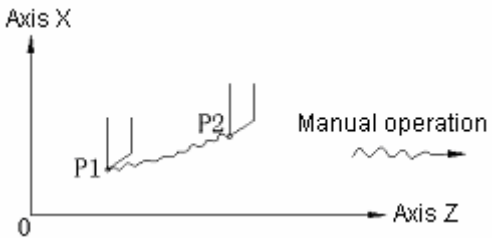
handwheel will differ from the travel distance of the machine. Hence do not rotate the handwheel too quickly.

Note 2: When ×100 override is selected and the handwheel is turned at quick speed or the workbench moves at “rapid traverse” rate, the machine will be subject to impact if it is stopped abruptly. The selection automatic acceleration/deceleration function is also valid for manual feed, thereby reducing mechanical shock.

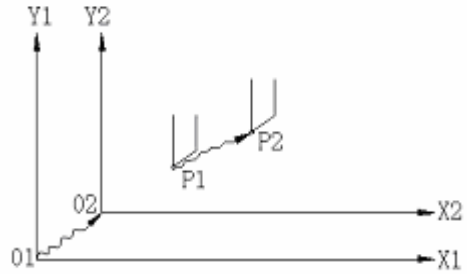
4.3.4.3 Manual absolute

If the switch is set to ON, the manual-operated travel amount will be added to the coordinate axes.

(1) MANUAL ABSOLUTE switch ON: Coordinates change with manual operation.



(2) MANUAL ABSOLUTE switch OFF: Coordinates do not change.



(Example) For example, in the following blocks:

…

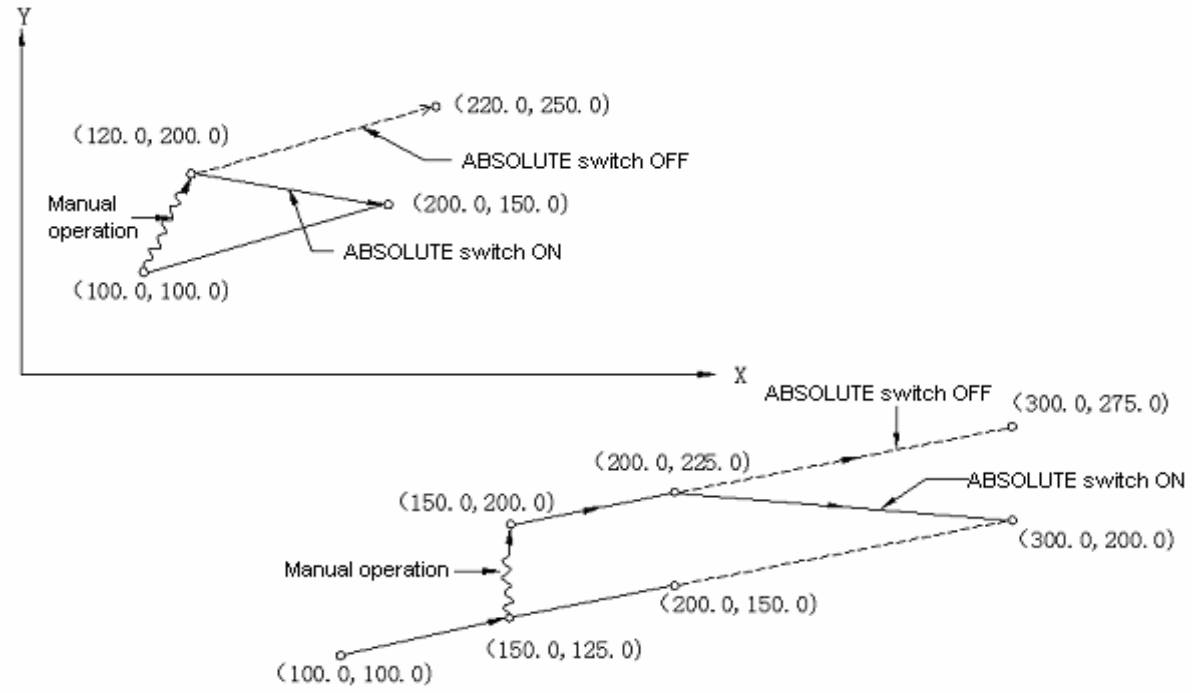
G01 G90 X100.0 Y100.0 F010；①

X200.0 Y150.0 ；②

X300.0 Y200.0 ；③

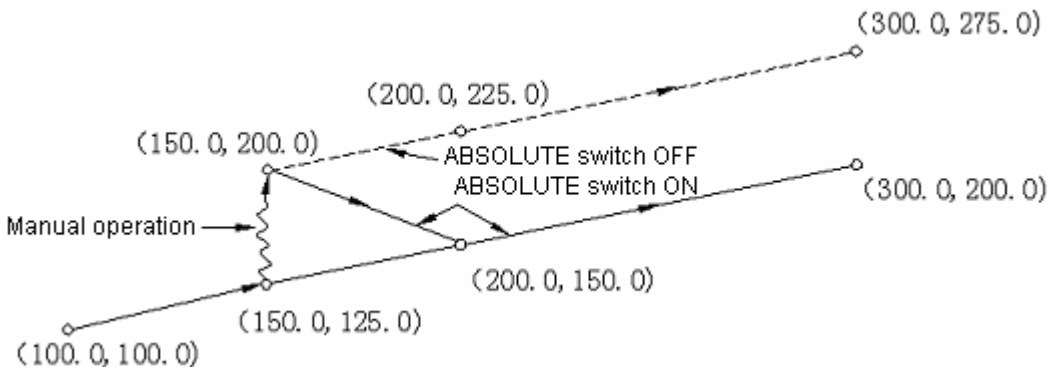
…

a) The above block ① has been executed while block ② is only executed after manual operation (travel by 20.0 in X direction and 100.0 in Y direction).



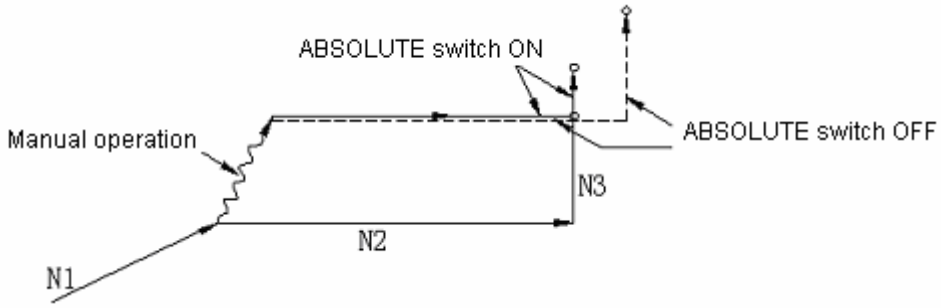
b) Press the FEED HOLD button in the execution of the block ②. After manual operation (Y+75.0), press the CYCLE START button so as to cancel the hold mode and continue the execution.

c) Press the FEED HOLD button in the execution of the block ②. Reset the machine after manual operation (Y+75.0). The block ② restarts inputting.



d) When manual operation is followed by a single-axis instruction, then only the instructed axis returns to the programmed absolute position of the axis.

```
N1  G01  G90  X100.0  Y100.0  F5000；
N2  X200.0；
N3  Y160.0；
```



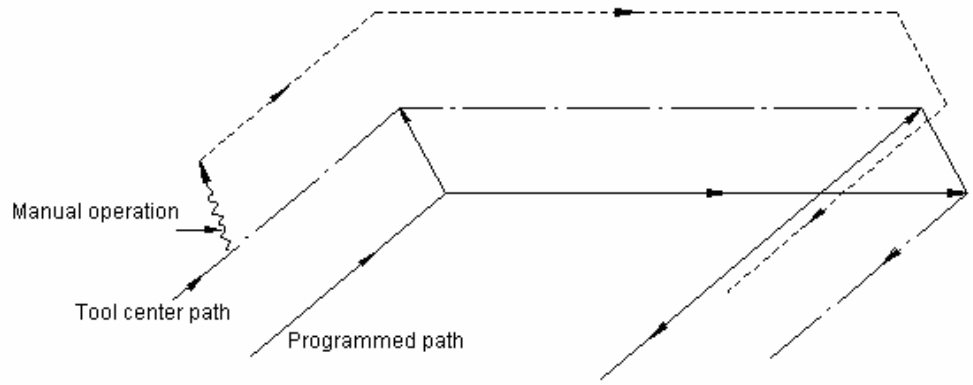
e) When manual operation is followed by an incremental instruction, then the position that the

axis moves to is identical with that is instructed while the MANUAL ABSOLUTE switch is set to OFF.

Note 1: Insert manual operations when tool radius compensation C offsets. Now the actual motion path of the tool is as follows:

(1) MANUAL ABSOLUTE switch OFF

When tool radius compensation C is enabled：If MANUAL ABSOLUTE switch is switched off for manual operation when tool radius compensation C is enabled, the path of the automatic tool motion will translate in parallel by the offset of the inserted manual operation.



(2) MANUAL ABSOLUTE switch ON

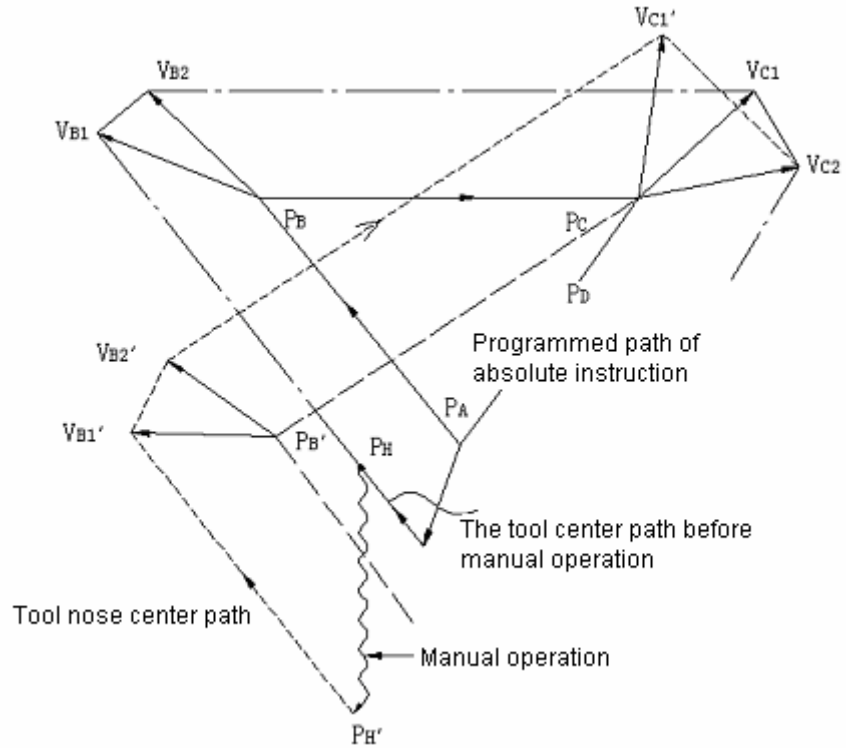
When tool radius compensation C is enabled：If MANUAL ABSOLUTE switch is switched on for manual operation when tool radius compensation C is enabled, the path of the tool under absolute instruction after restart is as follows. The tool path for the blocks after manual operation runs parallel to the vectors of the origin of the next block.

Tool path is determined by the vectors between the next block and the block that follow. For the angle machining with the intervention of manual operation, its tool path is identical with the above.

If a program consists of incremental instructions rather than absolute instructions, its tool path is identical with that when the MANUAL ABSOLUTE switch is set to OFF.

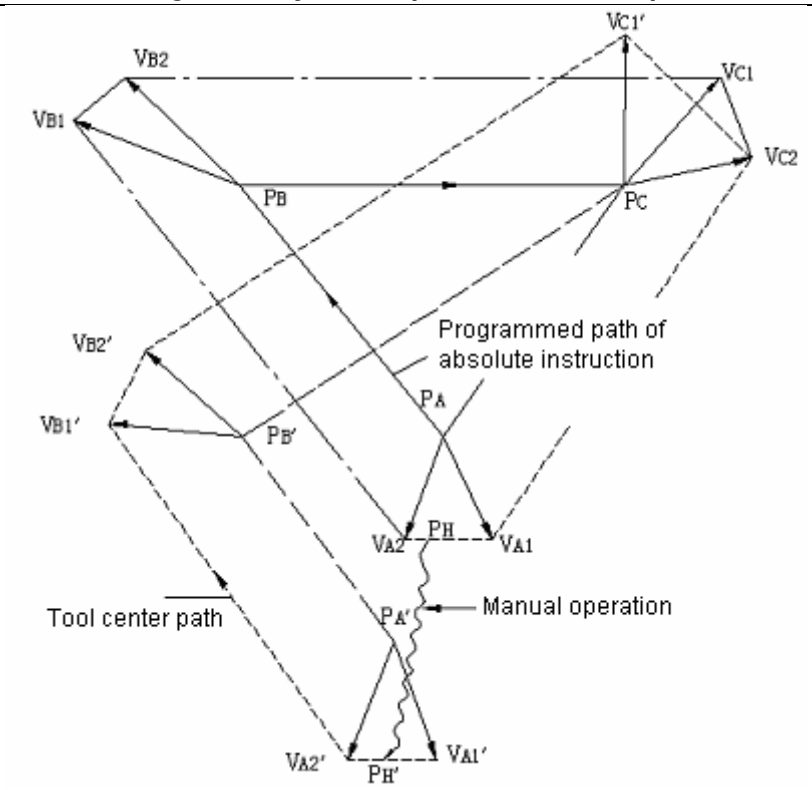
(a) Performing manual operations during execution of a block

Example 1: In the following programmed path (PA→PB→PC→PD), assuming the point PH between PA and PB is moved to point PH′ by manual operation after pressing the FEED HOLD button, the end point PB of the current block translates to point PB’ due to the offset as a result of manual operation and the vectors VB1 and VB2 of the original point PB also translate to V’B1 and V’B2.

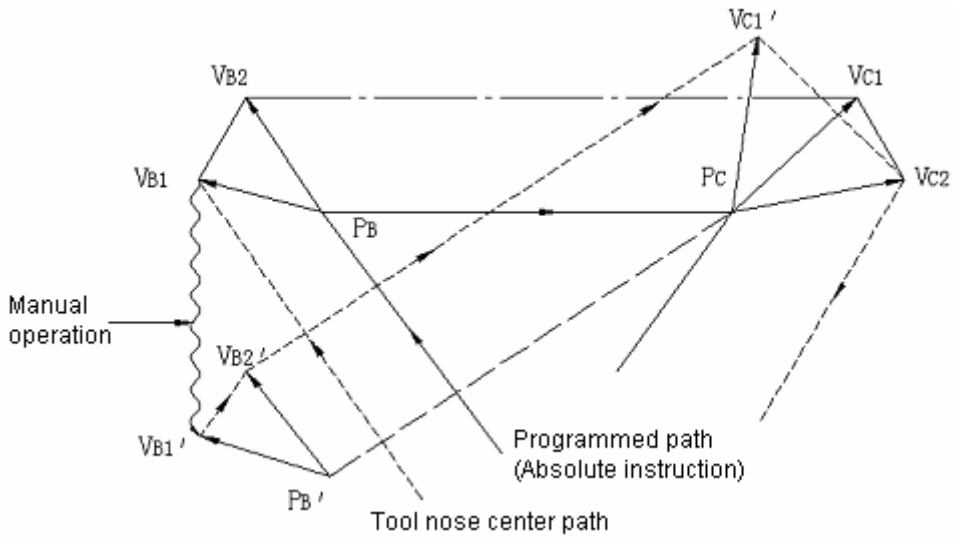


The vectors between the next block (tool path from PB to PC) and the one that follows (from PC to PD) do not need compensation. The new vectors (VC1′, VC2′) with compensation results from the relationship between the two blocks (programmed paths from PB′ and PC to PD and from PC to PD). Since vector VB2′ coincides with VB2, however, the section of path between PB' and PC as a result of tool offset is not accurately performed. But for the block after point PC, tool offset can be precisely performed.

Example 2: If manual operation is inserted in angle machining in the case of tool radius compensation, the feed path after manual operation will be determined by the same method as Example 1. That is, the vectors VA2′, VB1′ and VB2′ in the figure below are determined by translating the vectors VA2, VB1 and VB2 by an amount of manual travel and the new vectors result from V_{C1}' and V_{C2}'. The block after point PC will be precisely performed by the tool offset compensation C.



(b) If manual operation is inserted after the execution of single block function, the vectors VB1 and VB2 for the end points of the current block will be moved in parallel and the method for determining the following feed path will be identical with (a). MDI operation may be inserted after the execution of a block with single block function. The feed path after MDI operation coincides with tat after the insertion of a manual operation.



4.3.5 Manually returning to reference point (reference position)

The machine may return to the reference point by manual operations:

1) Set the SELECTION MODE to JOG.
2) Press the MACHINE ZERO key.
3) Move all axes toward the reference point by Jog feed.

   The machine rapidly traverses to the deceleration point and then to the reference point at FL

speed. Rapid traverse override is still active during traversing.

4) The machine stops at the reference point and the indicator indicating the end of the return to the reference point is lit.

Note 1: The indicator is lit after the return to the reference point. If the switch for returning to the reference point is set to ON position, the machine cannot translate in Jog mode.

Note 2: The following procedures may extinguish the indicator: (1) Move the machine away from the reference point; (2) Press the EMERGENCY STOP button.

Note 3: For the distance to the reference point, refer to the manual supplied by the manufacturer of the machine.

4.3.6 Operations on automatic running

The machine may automatically run by programs.

4.3.6.1 Start of automatic operation

Procedures for starting the program stored in memory:

(a) Select the program number. See Section 4.4.16 “Program number search”.

(b) Select AUTO mode.

(c) Press the CYCLE START button. Automatic operation starts once the CYCLE START button is pressed and at the same time the CYCLE START indicator is lit.

Note 1: The programs read in are loaded when the CYCLE START button is pressed in EDIT mode. The loading mode is the same as that when the INPUT button is pressed for parameter setting.

Note 2: The CYCLE START button is inactive in the following conditions：

(a) When the FEED HOLD button is pressed;

(b) When the EMERGENCY STOP button is pressed;

(c) When the RESET signal is enabled (contact the manufacturer of the machine for details);

(d) When the SELECTION MODE switch is set to a wrong position (other than AUTO or EDIT mode);

(e) When it is search a sequence number;

(f) When an alarm is given;

(g) When automatic operation is selected;

(h) When the NC system is not ready

4.3.6.2 Suspension of automatic operation

Press the FEED HOLD button

The FEED HOLD indicator illuminates and the CYCLE START indicator goes out when the FEED HOLD button is pressed. And now,

(a) If the machine is moving, the feed slows down and stops;

(b) If the machine is in hold state, the hold state will interrupt even in FEED HOLD mode;

(c) The machine stops after the performance of M, S, T or B function.

4.3.6.3 Single block

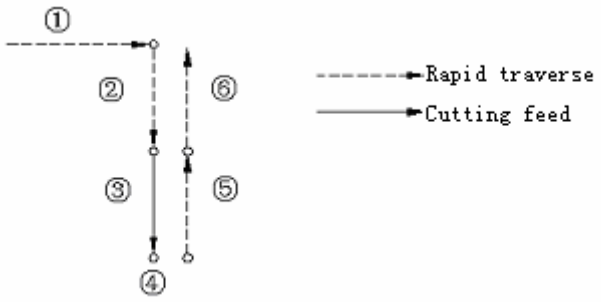
If the SINGLE switch is turned to ON position, the control only executes a block each time and

stops when the CYCLE START button is pressed.

The control only executes a block each time and stops when the SINGLE switch is turned on. When the CYCLE START button is pressed, the control stops after the execution of the next block.

Note 1: In G28, G29 or G30 mode, the tool stops at the intermediate point if the SINGLE function is used.

Note 2: For fixed circular processing, the tool stops at the end point of the circular path ①, ② or ⑥ of the fixed cycle (see the figure below) if the SINGLE function is active.



When the result of the fixed circular calculation is not 1, the FEED HOLD indicator illuminates except the block ⑥ of the final cycle. The FEED HOLD indicator illuminates whenever block ① or ② stops.

Note 3: For the blocks of M98P—, M99 and G65, G66 or G67, the stop of the single block is invalid. However, it is valid if the instructions in M98 or M99 block are of the addresses other than O, N, L and P.

4.3.6.4 Restart after feed hold or stop

(1) Select the AUTO mode;

(2) Press the CYCLE START button. The FEED HOLD indicator goes out when the CYCLE START button is pressed.

4.3.6.5 The manual operations in automatic run

(1) In automatic run, suspend the operation by pressing the FEED HOLD button on the operation panel or bring the SINGLE switch to ON position.

(2) Record the coordinates of the stop position displayed by the location display unit.

(3) Perform JOG operation (see Section 4.3.4.3).

(4) Return the tool to the recorded coordinates (the origin of JOG operation).

(5) Set the SELECTION MODE switch to the position before JOG operation so as to restart automatic run.

(6) Press the CYCLE START button.

4.3.6.6 The MDI operations in automatic run

(1) Set the SINGLE switch to ON position.

(2) Select MDI operation mode.

(3) Perform MDI operation.

(4) To restart automatic operation, return to the original operation mode and press the CYCLE START button on the operation panel.

Note 1: The modal data reserved in cycle movement is under influence when MDI instruction is used.

Note 2: The modal data instructed by MDI is still valid for automatic MDI operation.

Note 3: Too radius compensation C is not performed during MDI operation.

Note 4: MDI operation is disabled in feed hold state.

4.3.6.7 Skipping over optional blocks

When some block contains "/n"(n=1 to 9), the function allows the control to skip over the block. Switches correspond to the 19 numbers respectively.

Note: While blocks are being read in the buffer from storage, the validity of function of skipping over optional blocks is judged. Therefore the function is invalid for the blocks read in buffer register.

4.3.6.8 Feedrate override

For the feedrate set by F function, it is possible to set an override in the range of 10% to 200%. An override of 10% increment is recommended.

4.3.6.9 Dry run

If the switch is set to ON position in the cycle operation instructed by storage or MDI, the F function does not work and the machine travels at the following speeds.

RAPID button ON/OFF	In rapid traverse	In cutting feed
RAPID button ON	Rapid traverse	Maximum Jog feedrate
RAPID button OFF	Jog feedrate (see Note)	Jog feedrate

Note: The dry run of rapid traverse may be disabled or enabled by parameter setting.

4.3.6.10 Machine lock

When the MACHINE LOCK switch is set to ON, the movement instruction pulse is inhibited. Therefore the position indication for circular feed start or JOG operation is continuously updated according to input instructions. But the machine does not move itself. The function is used to check procedures.

Note 1: When G27, G28 or G30 instruction is set, the machine will not return to the reference point. Hence the indicator for returning to the reference point is not lit.

Note 2: M, S, T and B functions are performed.

4.3.6.11 Display lock

When the DISPLAY LOCK switch is activated, the coordinates indicated by the location display unit are locked. For instance, when the coordinate system is moved as a result of JOG operation, the use of this switch prevents the indicated values from changing by manual movement.

Note: The function is optional.

4.3.6.12 Mirror image

Once the mirror image switches of axes X and Y as well as the 4^{th} axis are activated in automatic operation, the axes move reversely. The reference point is returned to by JOG or automatic operation, the movement between the intermediate point and the reference point does not inverse and position display depends on the actual movement of the tool. This may be achieved by setting parameters with MDI unit (see Section 4.4.7).

4.3.6.13 Rapid traverse override

It is possible to set the rapid traverse override switch of optional overrides 100%, 50%, 20% and F0 on the operation panel of the machine.

When the feed speed is 10m/min and the switch is set to 50% position, the actual feed speed will be 5m/min.

F0 is a fixed speed (feedrate) provided by the manufacturer of the machine. The function applies to the following conditions:

(1) The rapid traverse specified by G00;

(2) The rapid traverse in fixed cycles;

(3) The rapid traverse in G27, G28, G29 and G30 modes;

(4) JOG rapid traverse;

(5) The rapid traverse for manually returning to the reference point.

4.3.6.14 External workpiece number search function

Select a workpiece number to be processed with the switch on the operation panel of the machine (No example of the operation panel is given in this user manual). (Machining programs are pre-stored in the part program storage.) Press the START button. Now the system automatically executes the program corresponding to the workpiece number. By using the function, operator does not need to search the stored program numbers so as to reduce dry run time and errors.

(1) Preparation for the program: In the situations using the function, the numbers assigned to programs shall correspond to the workpieces to be processed. That is, a number among 01 to 31 shall be designated for each workpiece to be processed. The relevant program number is expressed as follows:

0(:) 0 0 □ □ (0 for EIA and : for ISO)

Workpiece number (01 to 31)
Optional workpiece number

They are stored in the part program memory. As shown in the following examples, each program shall be started by the address 0 followed by a program number and ended by M02, M30 or M99.

In addition, the storage of the programs irrespective of workpiece number is allowable.

```
0    0001;
N    0001 G00…;          The program corresponding to workpiece No. 01
……
……

N    120 M02;
0    0002;
N    0001 G00…;          The program corresponding to workpiece No. 02
……
……
N    300 M30;
0    0004;
N    001 G00……
……
                         The program corresponding to workpiece No. 04
……
```

N 080 M02;

0 6247;
N 001 G00……
……
……
N 034 M99;

} Programs irrespective of workpiece number

Note 1: Each program shall be started by the address 0 followed by a program number and ended by M02, M30 or M99. However, M02, M30 and M99 cannot be specified in the middle of the program. If one of them is specified in the middle section, the program that follows will regarded as another program segment (the block following M02, M30 or M99 is immediately numbered as a program when the program is stored in memory).

Note 2: The allowable quantity of workpiece numbers depends on factory setting (see the manual of the machine).

Note 3: For the machine system provided with external workpiece number search function A, the allowable maximum workpiece number is 31. Now the first two digits of the program number corresponding to a workpiece number must be 00.

(1) Operating procedures

Operating procedures vary with different manufacturers of machine. The operating procedures described below are general. Refer to the manual supplied by manufacturer of the machine for specific operating procedures.

Note 1: Select the automatic mode and then set the program (01 to 31) corresponding to the workpiece number with the rotary switch on the operation panel on the machine side. When the START button is pressed, the program corresponding to the set workpiece number will be searched out and machining performed with the start of the program.

Note 2: When a workpiece number is set to 00, the corresponding program will not be searched if the START button is pressed. The execution starts from executable section of the current program. For the situations that starts in the midway of the program or that the executing program is independent of workpiece number, it is necessary to set the workpiece number to 00 and press the START button after sequence number search or program number search.

Note 3: The function does not apply to MDI operation but automatic operation.

Note 4: If a program number corresponding to the workpiece number is not stored in the memory, an alarm (No. 59) will be given once the START button is pressed.

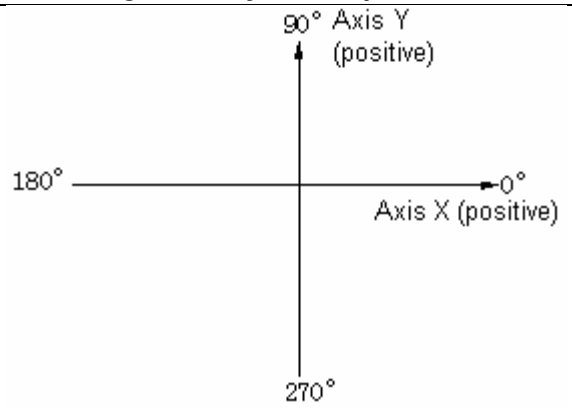
Note 5: It is not always necessary to select the relevant program even a workpiece number is selected with the dial. Refer to the manual supplied with the machine for the procedures for selecting a program. When workpiece number search function A is selected, program search is performed after the NC system starts automatic operation in reset mode.

4.3.7 JOG feed at any angle

Set an angle and feedrate in the plane of X and Y and then press the START button. In this way the machine may feed at any set angle by JOG operation.

(1) Set the SELECTION MODE switch to the mode of JOG feed at any angle

(2) Set an angle with the angle setting dial. The position of an angle is selected among 0-71 with a 2-digit BCD code. 0 ~71 correspond to 0 ~360° respectively (in 5° increment).



For angle setting, make sure to switch on angle strobe pulse. If angle strobe pulse is switched on, the formerly set angle will remain valid.

As shown in the above figure, the + direction of Axis X is 0° and that of Axis Y is 90°.

(3) Select a feedrate (speed in tangential direction) with the Jog feed dial.

(4) Press the START button in the mode of JOG feed at any angle. Then the machine moves at the selected feedrate in the set direction.

If the JOG RAPID button is pressed, the machine will feed at the maximum Jog feedrate. The machine feeds when the JOG RAPID TRAVERSE button is switched off and stops feed when it is switched off.

Note 1: If axes X and Y are interlocked, both axes will slow down and stop. They will restart once the interlocking is disabled.

Note 2: In automatic operation, it is possible to insert JOG feed at any angle when the machine is stop in feed hold mode.

Note 3: For the situation with external deceleration selection, JOG feed at any angle is also active. Now the tangential feed is equal to the external decelerating rate.

Note 4: The automatic acceleration and deceleration for cutting feed also apply to the JOG feed at any angle.

Note 5: JOG feedrate at any angle does not change with Jog feedrate even during the switching between metric and inch systems.

4.3.8 Manual insertion

For the specific axis (fixed by parameter) in automatic operation, the movements operated with handwheel may be performed in addition to the self-motion of the axis.

4.3.8.1 Inserting operations by MPG(manual pulse generator/handwheel)

Manual insertion is possible by turning the manual pulse generator in the following conditions.

(1) Mode: automatic mode or MDI mode

(2) Operating state: Manual insertion is possible during linear interpolation, arc interpolation, spiral interpolation or sine-curve interpolation.

However, the following conditions are excluded:

(I) When an alarm is given;

(II) When any axis does not move;

(III) When positioning is valid;

(IV) When interlocking is active;

(V) In the absence of travel instruction.

(3) Manual axis selection signal

Manual axis selection signals (HX, HY, HZ, H4 and H5) are switched on (contacts close) for the axes to perform manual insertion.

4.3.8.2 Manual inserting movement by MPG

(1) Amount of travel: The amount of travel to be inserted by manual shall be identical with that during JOG feed. The amount of travel depends on the scale of the manual pulse generator and JOG feed overrides (X1, X10 and X100) and is added to that of automatic operation.

(2) Traverse speed: The axial speed for JOG insertion is the result of the addition of the travel speed of automatic operation to that inserted by manual. Therefore, axial speed is limited to rapid traverse speed (Parameter HR) in the event that axial speed exceeds rapid traverse speed. Displacement mismatches the indicated value of the manual pulse generator.

(3) The correspondence between manual-inserted travel and all signals is as follows:

Signal	Travel
Machine is locked	Affected: The tool does not move when MACHINE LOCK is enabled.
Display is locked.	Affected: Relative coordinates remain unchanged when display is locked.
Mirror image of Axis X	Not affected: The machine moves forward when the handwheel is turned clockwise.

(4) The correspondence between manual-inserted travel and position indications is as follows:

Indication	Travel
Absolute coordinates	Not affected: Manual-inserted pulse is not added to absolute coordinates
Relative coordinates	Affected: Manual-inserted pulse is added to relative coordinates
Mechanical coordinates	Affected: Manual-inserted pulse is not added to mechanical coordinates

(5) Indication of amount of travel: Manual-inserted amount of travel may be displayed in diagnosis message (Diagnosis No. 805 to 809). To display a diagnosis message, press the function key DIAGNOSIS on the MDI panel.

Diagnosis data numbering

8 0 5 Manual-inserted amount of travel of Axis X

8 0 6 Manual-inserted amount of travel of Axis Y

8 0 7 Manual-inserted amount of travel of Axis Z

8 0 8 Manual-inserted amount of travel of the 4^{th} axis

8 0 9 Manual-inserted amount of travel of the 5^{th} axis

Unit: 0.001mm (input in metric system)

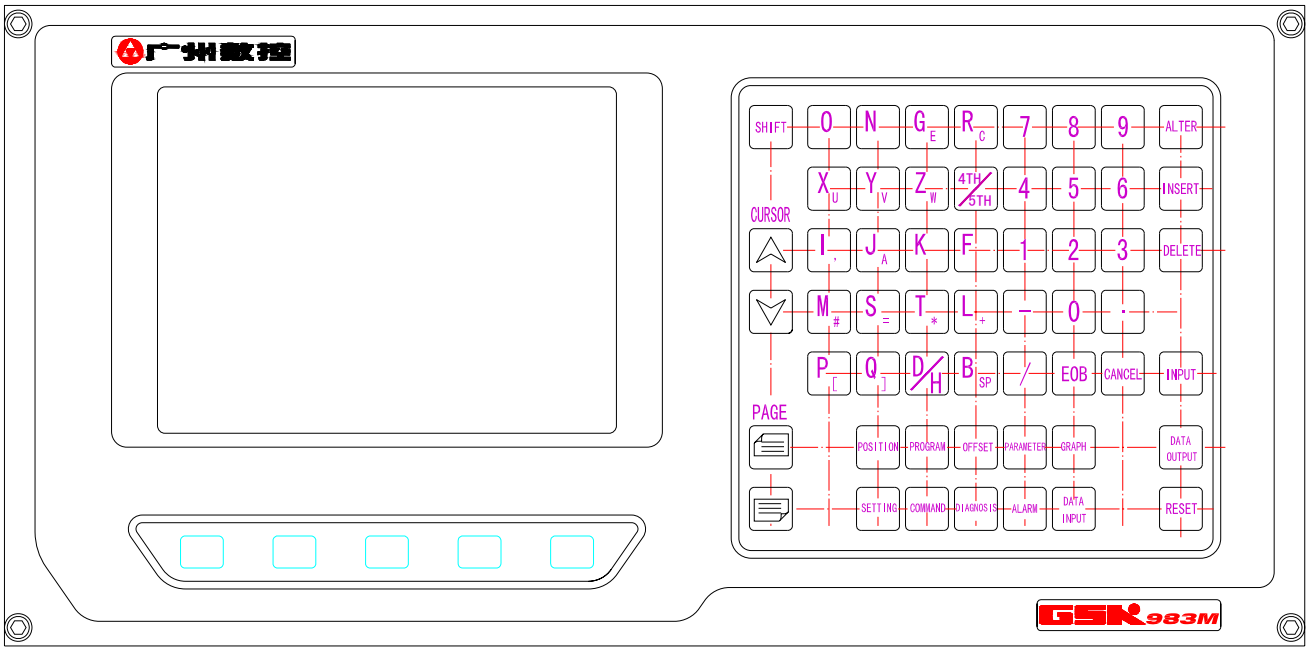
0.0001 inch (input in inch system)

Note: Only the removable amounts of travel are cleared.

4.4 The Display and Operations of the MDI/LCD Panel with LCD Characters

The MDI/LCD panel is usually mounted on the upper front side of the control cabinet. It consists of an LCD and buttons as follows.

Function buttons: The large number of items displayed with the function buttons is just like the chapters of a book. When a function button is pressed for the second time and third time, the chapter 2 or 3 of the corresponding display functions (if the function button for the chapter is provided). Each chapter includes several pages and each page is selected with the page button.



The names and meanings of all function buttons are listed below.

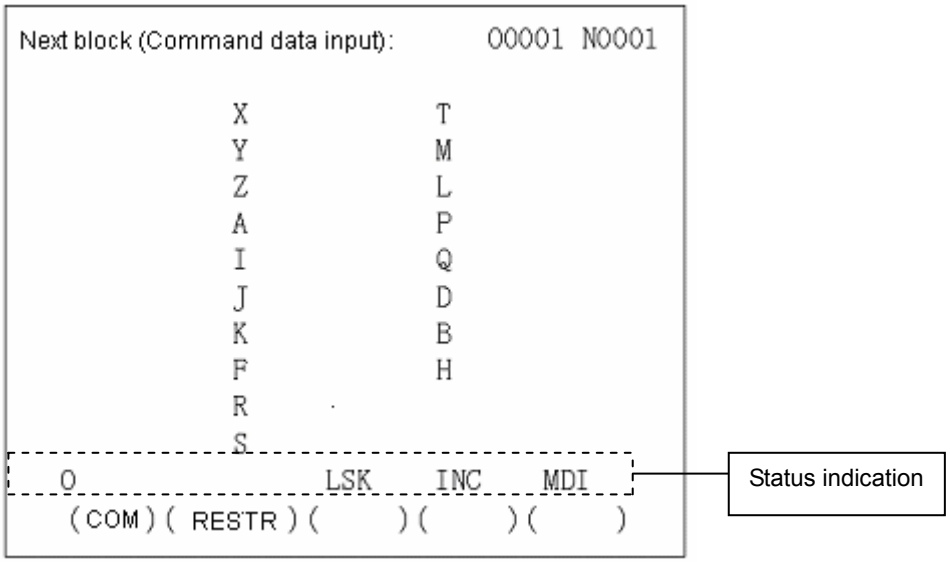
POSITION	Pressing once	Display of actual position and reset
SETTING	Pressing once	Display and setting of set data
	Pressing twice	Display and setting of user macro variables
	Pressing for the third time	Display and setting JOG switch
PROGRAM	Pressing once	Display of the information regarding a program in EDIT mode Display of the executing or executed blocks and the blocks that follow in a mode other than EDIT
	Pressing twice	Display of the list of program numbers (See Section 4.4.24.12) (The chapter 2 may also be omitted depending on the conditions of the system.)
PARAMETER	Pressing once	Display and setting of parameters
	Pressing twice	Display and setting of PC parameters
OFFSET	Pressing once	Display and setting of offset
	Pressing twice	Display and setting of origin offset in a workpiece coordinate system
ALARM	Pressing once	Display of the information of an alarm
	Pressing twice	Display of an external alarm and external information
COMMAND	Pressing once	Display of instruction value and the instructions input through MDI
	Pressing twice	Display of the information regarding program restart

DIAGNOSIS	Pressing once	Display of system diagnostic data
	Pressing twice	Display of the information regarding tool life management

Note: Clear the displayed screen by concurrently pressing a function key and CANCEL button. The corresponding screen is displayed when the function button is pressed again.

4.4.1 Status display

The status indication of the system is displayed on the lower right part of the screen:

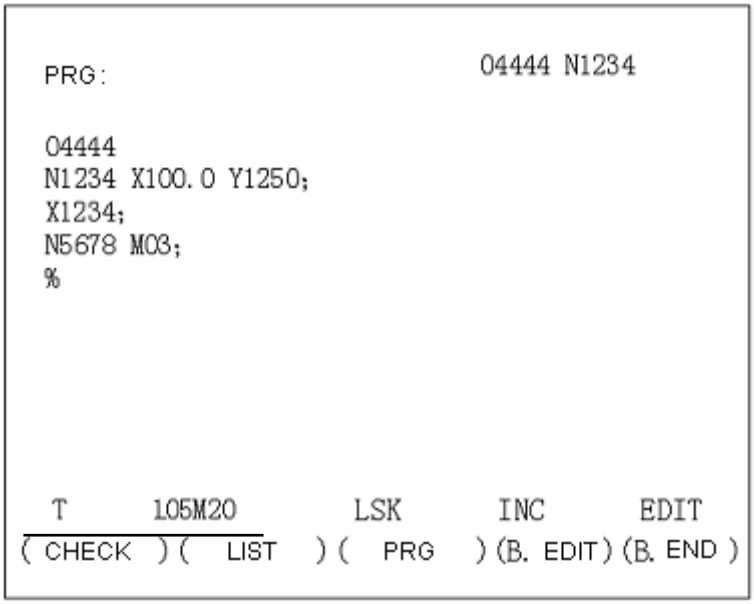


The displayed indications are as follows:

NOT READY indicates that the control or servo system fails to operate. LSK indicate the LABEL SKIP mode created after power on or reset of control rather than in MDI mode. BUF indicates that a block is read in but not executed. The block not executed still does not disappear after reset in rather than MDI mode. ABS indicates that MDI instruction is absolute and INC state will be entered into when the SHIFT button is pressed. INC indicates that MDI instruction is incremental ABS state will be entered into when the SHIFT button is pressed. ALM indicates that an alarm is given. The alarm type will be displayed (the symbol blinks) when the ALARM button is pressed. EDIT indicates that the editing function is being executed (the symbol blinks). The stopping operation of edition shall be performed when the symbol exits. SRCH indicates that sequence search is being performed (the symbol blinks). RESTR indicates that the period from program restart to the return to the final axis (the symbol blinks).

4.4.2 Key input

The entries input with address keys or numerical keys are displayed at the bottom of the screen.



Data cannot be typed in any more when the POSITION or and ALARM button among the function buttons is pressed to display a screen.

Press D/H to enter D and again to enter H.

Only a word consisting of one address and a figure can be typed in when Program edit is not being performed. Pressing CANCEL once clear a word.

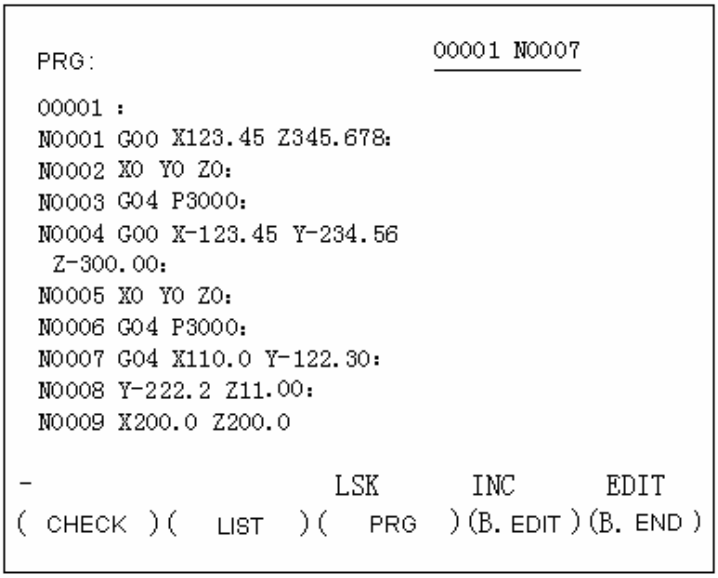
One or more words, a block or any character string of up to 32 characters can be entered with the keys during Program edit.

The last entered character is cleared by pressing the CANCEL key. If the CANCEL key is pressed continuously, the typed characters will be cleared in succession.

Note: In EDIT mode, Program edit is possible when the PROGRAM button is pressed.

4.4.3 Display of program numbers and sequence numbers

Program numbers and sequence numbers are displayed at the top of the screen as shown in the following picture.



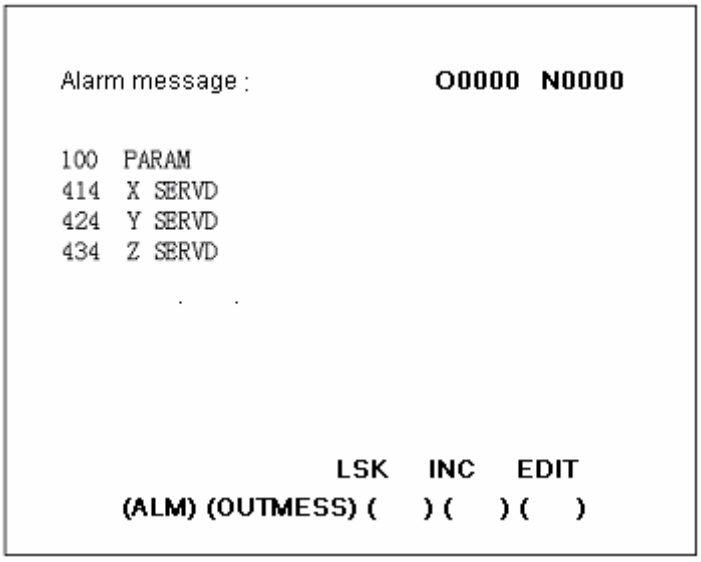
The meanings of the displayed sequence numbers and program numbers are as follows:

Mode	Operation	Indication
Mode other than EDIT	In the mode other than EDIT	To display the last displayed sequence number
	When search a sequence number	To always display the sequence number during search
Auto mode (MEMORY)	Pressing the cursor ↑ key when the function button is in PROGRAM mode	To return to the start of a block To display the block
Edit mode	Pressing the cursor ↓ key when the function button is in PRG mode	To review programs in + direction from the actual position of the storage; To display the firstly found N value
	Pressing the cursor ↑ key when the function button is in PRG mode	To review programs in - direction from the actual position of the storage; To display the firstly found N value
	Entering reset state by pressing the RESET button	To return to the switch of the block and display the block
Auto mode (MEMORY)	Program number search	To display the program numbers searched

4.4.4 Alarm display (function button ALARM)

When ALM is indicated at right bottom of the screen in case of alarm, clear warning messages through the following procedures
Press the ALARM button. When the information about operator is displayed, press the ALARM button again to display alarm message.

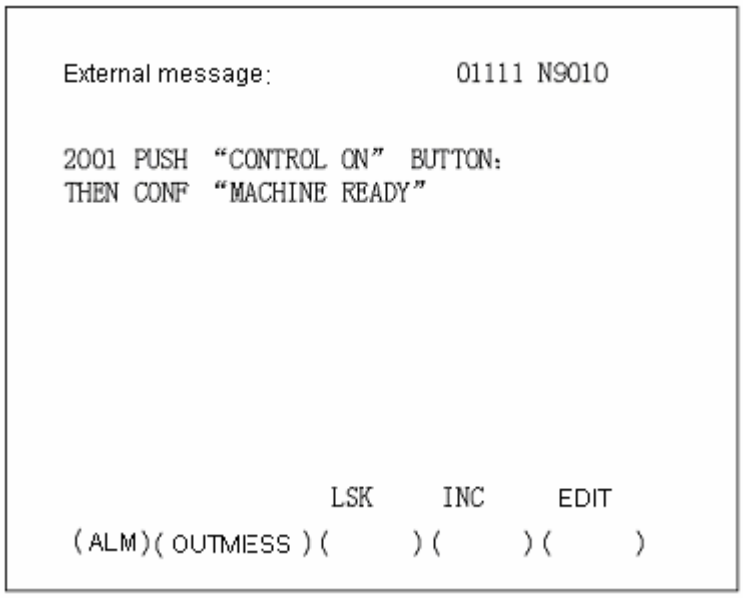
Refer to Appendix 6 for the meanings of all alarm numbers



Note: As a rule, alarm message appear on the screen in the event of alarm.

4.4.5 Operator information

Once the machine sends out operator information, the information will be automatically displayed on the screen.



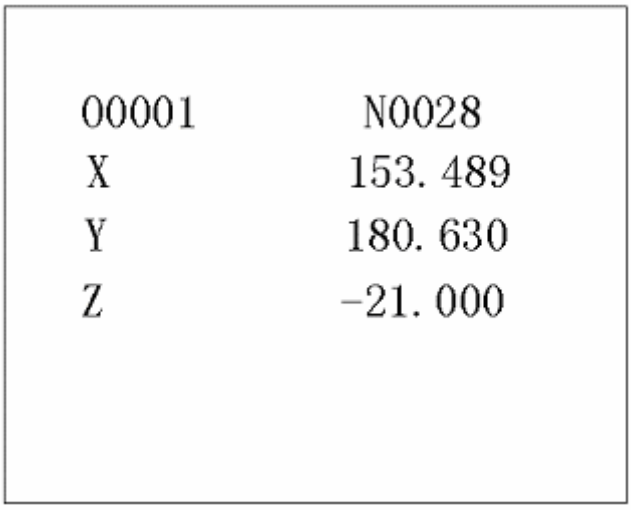
When operator information appears after some other page is displayed, press the ALARM button. When alarm message appears, press it again.

4.4.6 Display of actual position and reset (function key POSITION)

(1) Press the POSITION button.

(2) Press the Page button. Data is displayed in one of the following three modes.

(I) Position display in a relative coordinate system



Relative position is displayed once operator resets a position to zero.

Reset: When the X, Y, Z or 4TH/5TH button is pressed, the pressed address characters will continuously blink. The relative position of the blinking address will be set for 0 when the SHIFT button is pressed again.

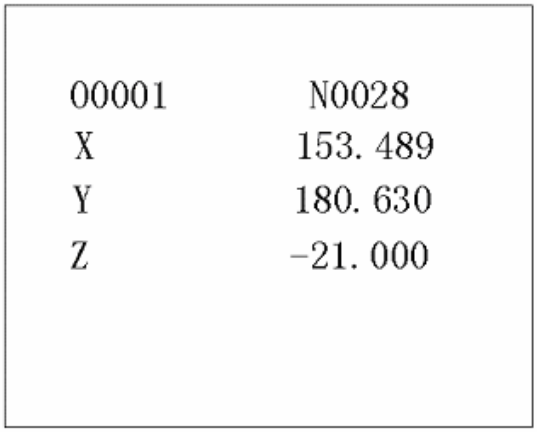
(II) Position display in a workpiece coordinate system

The current value of program coordinate system is set by G92, automatic coordinate system or reset and displayed. For Axis T, the currently selected tool number is displayed. Reset

(program protection unlocking)

For resetting, the [X], [Y], [Z] or [4TH/5TH] button is pressed. The pressed address characters will continuously blink. The coordinate position of the blinking address when the [SHIFT] button is pressed again. The actual position of the blinking address is reset.

Note: Reset operation can only be performed in automatic stop status.



(III) Display of comprehensive position

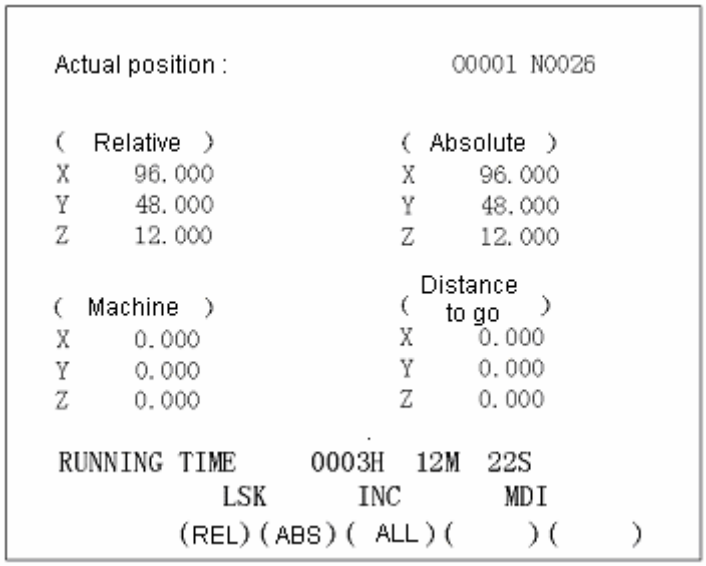
The actual position is displayed in the following coordinate systems at the same time:

(a) The position in relative coordinate system (RELATIVE)

(b) The position in absolute coordinate system (ABSOLUTE)

(c) The position in machine coordinate system (MACHINE)

(d) The distance to be traveled (DISTANCE TO GO)



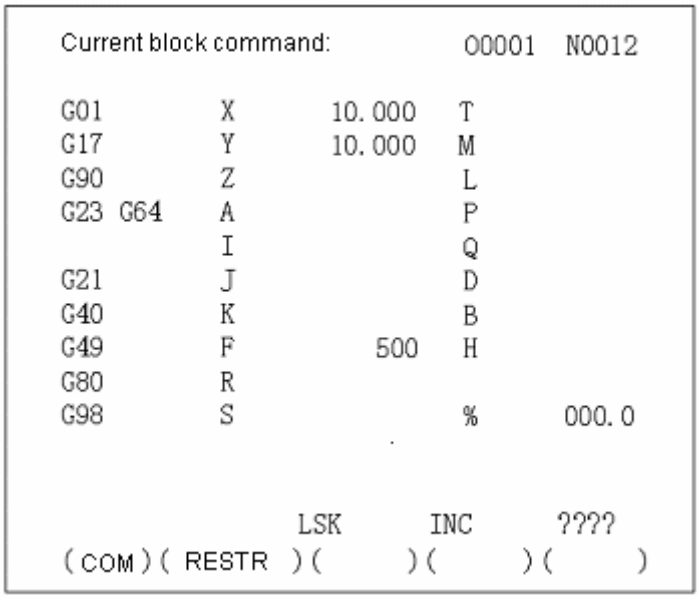
DISTANCE TO GO indicates the remaining distance of a block. The positions of all coordinate systems cannot be reset when displaying the comprehensive position. The unit of machine coordinate system is identical with that of the machine system.

4.4.7 Indication of instruction value (function button [COMMAND])

(1) Press the COMMAND button.

(2) Press the PAGE button. Data is displayed in the following two modes.

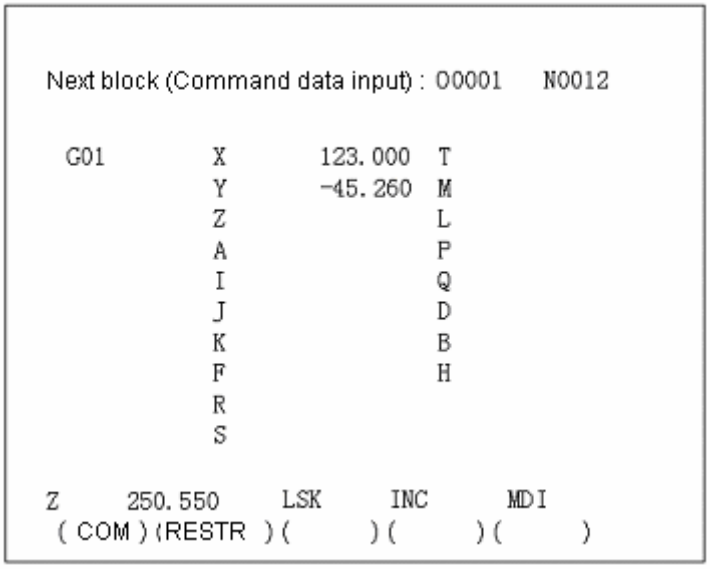
(I) Display the formerly set modal values while executing an instruction value.

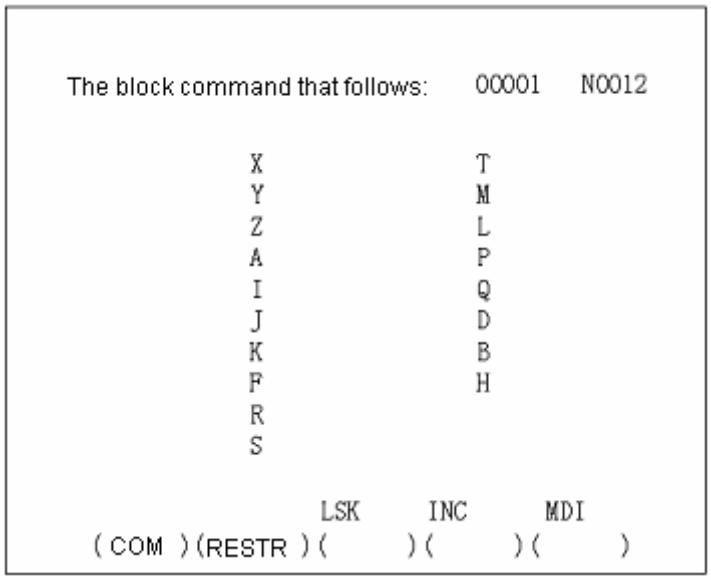


As shown in the above figure, the figure following character % stands for feedrate.

(II) Display the instruction value input by MDI or the instruction value to be executed next time.

(III) Display the instruction value of the next block to be executed during the tool offset of tool radius compensation C.





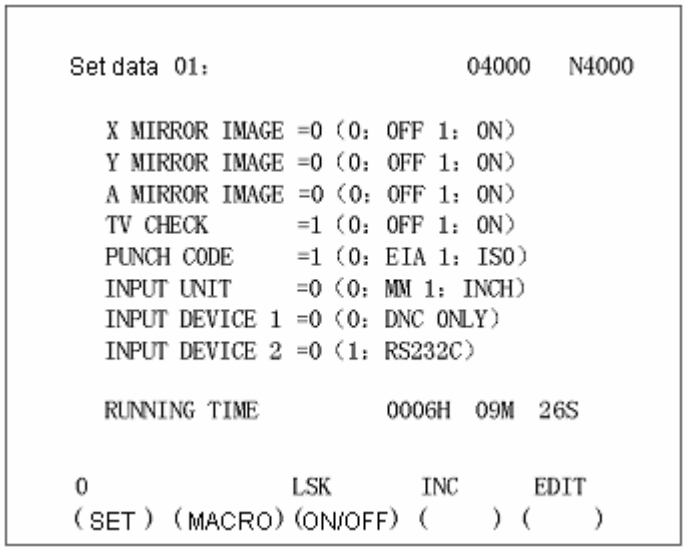
4.4.8 Setting (function button SETTING)

4.4.8.1 Display and setting of input, output, etc

(1) Press the SETTING button.

(2) Press the PAGE button. Setting and display may be performed in the following two modes.

(I) Setting of input and output


Set data 01: 04000 N4000
X MIRROR IMAGE =0 (0: OFF 1: ON)
Y MIRROR IMAGE =0 (0: OFF 1: ON)
A MIRROR IMAGE =0 (0: OFF 1: ON)
TV CHECK =1 (0: OFF 1: ON)
PUNCH CODE =1 (0: EIA 1: ISO)
INPUT UNIT =0 (0: MM 1: INCH)
INPUT DEVICE 1 =0 (0: DNC ONLY)
INPUT DEVICE 2 =0 (1: RS232C)
RUNNING TIME 0006H 09M 26S
0 LSK INC EDIT
(SET) (MACRO) (ON/OFF) ( ) ( )

Setting (active when the program protection lock is disabled and inactive when it is locked, which can be switched by parameter)

(a) Set the SELECTION MODE switch to MDI mode.

(b) Press the cursor button to move the cursor to the position of the item to be changed. The cursor cannot be moved with the address key N.

(c) Enter 1 or 0 with the P key as shown in the following table.

	0	1
X MIRROR IMAGE	MIRROR IMAGE OFF	MIRROR IMAGE ON
Y MIRROR IMAGE	MIRROR IMAGE OFF	MIRROR IMAGE ON
4th AXIS MIRROR IMAGE	MIRROR IMAGE OFF	MIRROR IMAGE ON
TV CHECK	NO	YES
PUNCH CODE	EIA	ISO
INPUT UNIT	mm	inch
INPUT DEVICE1	DNC (can only be set to 0)	
INPUT DEVICE2	Unused	RS232C input

Press [P], [O] or [1], [INPUT] to proceed.

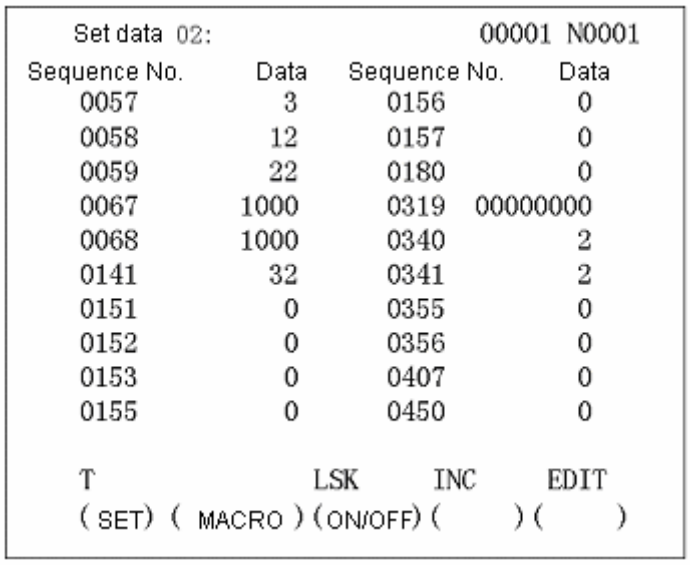
Note 1: Unselected selection function cannot be set. For example, INPUT UNIT=1 cannot be used for a metric-system machine when metric/inch system selection function is not available. PUNCH CODE=1 cannot be set when ISO code input selection function is not available.

Note 2: INPUT UNIT is automatically rewritten when executing G20 (input in inch system and G21 (input in metric system).

Note 3: The ISO or EIA specified by PUNCH CODE is independent of input during data output. ISO or EIA code can be automatically identified.

Note 4: The output device for data output is set with data No. 341.

(II) Other settings and indications



The displayed numbers and their meanings are as follows:

Data No.	Meaning
057	Run time (Unit: hr) (TMHOR)
058	Run time (Unit: min) (TMMIN)
059	Run time (Unit: sec) (TMSEC)
067	The retraction (CYCR) in fixed cycle G73 (depth high-speed Jog touring cycle)
068	The cutting origin in fixed cycle G83 (depth Jog touring cycle)
141	Run time (TIMDE1)
151	X value of Acme 1 of storage type travel limit 2
152	Y value of Acme 1 of storage type travel limit 2
153	Z value of Acme 1 of storage type travel limit 2
155	X value of Acme 2 of storage type travel limit 2
156	Y value of Acme 2 of storage type travel limit 2
157	Z value of Acme 2 of storage type travel limit 2
180	The sequence number whose execution has stopped
319	Settings (PRG8.MSBL)

340	Input device for selecting data storage (IDVICE)
341	Output device for selecting data for output (ODVICE)
355	Decelerating distance (automatic angle override) of the end point of block
356	Decelerating distance (automatic angle override) of the origin of block
407	Scaling override

Note 1: The data numbers other than those listed in the above table are not displayed.

Note 2: It is also possible to set the data number identical with the above table as a reference number.

Note 3: Refer to Appendix 5 for the details about data numbering.

Note 4: The details of data No. 340 and 341 are as follows:

3 4 0	I DVICE
3 4 1	O DVICE

IDVICE is used to select an input device for storing data in memory. When the set input device (INPUT DEVICE)2=1(Interface RS232), the setting is valid.

ODVICE is used to select an output device for data output.

Setting	I/O
0	Input: paper tape reader; output: FACIT PUNCHER
1	Common for input and output: ASR33/ASR43; Set baud rate and other parameters to 310.
2	Common for input and output: reader/puncher; Set baud rate and other parameters to 311.
3	Common for input and output: reader/puncher; Set baud rate and other parameters to 312.
4	Common for input and output: reader/puncher; Set baud rate and other parameters to 313.

It is also possible to set them by parameters.

Setting (active when the program protection lock is disabled and inactive when it is locked, which can be switched by parameter)

(a) Set the SELECTION MODE switch to MDI mode.

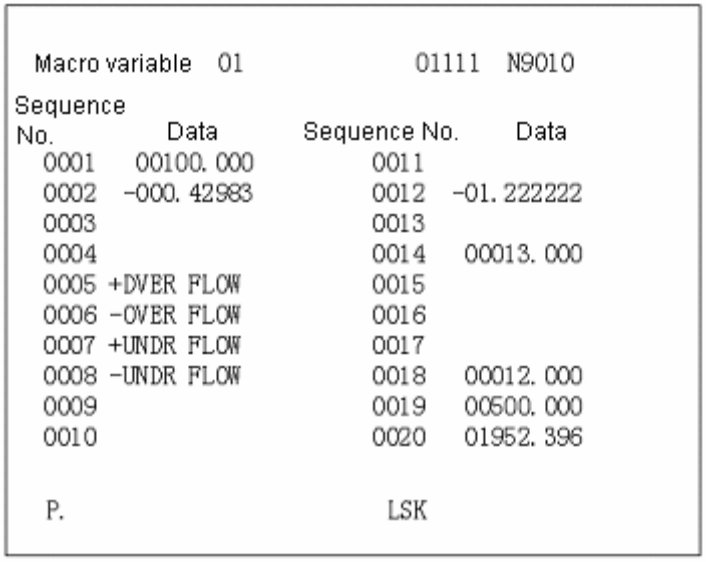
(b) Press the cursor button to move the cursor to the position of the item to be changed. The cursor cannot be moved with the address key N.

(c) Press P, numerical keys and INPUT in succession to proceed.

4.4.8.2 Display and setting of user macro variables

It is possible to display general variable values and the local variable values of the currently called user macro body on LCD.

When a variable value is <Empty>, the display will be blank. When an absolute value exceeds 99999999, it displays OVER FLOW. When an absolute value is not 0 but less than 0.0000001, it displays UNDER FLOW.



Display

(1) Select a set chapter 2

Press the SETTING button for SETTING DISPLAY and press it again.

(2) Since the display covers 6 pages, you need to press the PAGE button to display the required page.

Page 1——The currently called local variables #1-#20 for nesting

Page 2——The currently called local variables #21-#33 for nesting

Page 3——General variables #100-#119

Page 4——General variables #120-#139

Page 5——General variables #140-#149

Page 6——General variables #500-#509

(3) Move the cursor to the variable number to be displayed.

Method 1: Press the cursor button and move the cursor in succession. The next page will be switched to once the cursor goes beyond the current page.

Method 2: Set by typing with N, variable number and INPUT (active when the program protection lock is disabled).

(a) Select MDI mode;

(b) Type with P, variable number and INPUT when the variable is displayed and the cursor is moved to the variable number to be changed.

4.4.9 Operating through MDI (function key COMMAND)

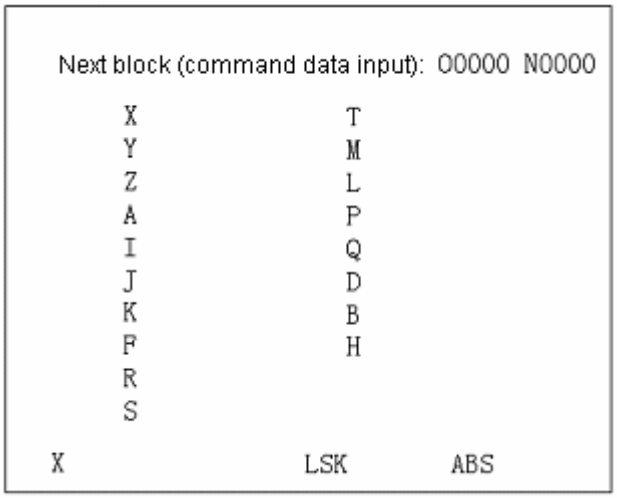
A program instruction to be executed may be input through MDI&DPL panel.

(1) Example: X 10.5 Y200.5

(a) Set the selector switch to MDI position.

(b) Press the COMMAND button.

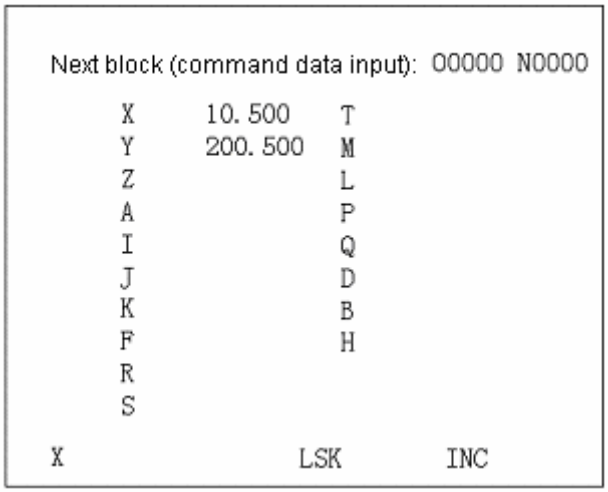
(c) Press the PAGE button. “Next block (command data input)” appears on the upper left of the screen.



(d) Press the [X], [1], [0], [•] and [5] keys and the [INPUT] button in succession. If the numeral entered before pressing the [INPUT] button is incorrect, press the [CANCEL] button and enter the correct numeral again. If an error is found after pressing the [INPUT] button, it is necessary to enter the numeral again.

(e) Press the [Y], [2], [0], [0], [•] and [5] keys and the [INPUT] button. If the typed numeral is found incorrect, proceed in the same way as inputting X.

(f) Press the CYCLE START button on the control panel of the machine.



(2) Delete Y200.5 from X10.5 Y200.5 before pressing the CYCLE START button.

(a) Press the [Y], [CANCEL] and [INPUT] in succession.

(b) Press the CYCLE START button on the control panel.

(3) Delete modal data.

Since G code and F, D and H data cannot be deleted, it is necessary to input correct modal data again for modification.

4.4.10 Start of MDI motion

Press the CYCLE START button to execute the instruction input through MDI.

4.4.11 Reset

As a rule, pressing the [RESET] button cancels alarm state.

Once the [RESET] button is pressed, the NC system is set to the following states:

State before reset		State after reset
In execution of a move instruction		The tool slows down and stops and the remaining travel disappears.
M, S, T or B In transfer		The transfer sequence stops. Refer to the manual supplied with the machine for the state of the machine now.
Storage 1 of buffer memory, block	MDI mode	The contents of the buffer memory are not eliminated.
	Mode other than MDI mode	The contents of the buffer memory are eliminated and the BUF label disappears.

In any case, pressing the RESET button sets the NC system to reset state. In the modes other than MDI, the NC system is set to the LABEL SKIP mode.

4.4.12 Tool position offset

The setting display of tool radius compensation (function button: OFFSET)

(1) Press the OFFSET button.

(2) Press the PAGE button and display the required page.

Position offset No. 1-12 of page 1;

Position offset No. 13-24 of page 2;

Position offset No. 25-32 or 25-36 (optional) of page 3;

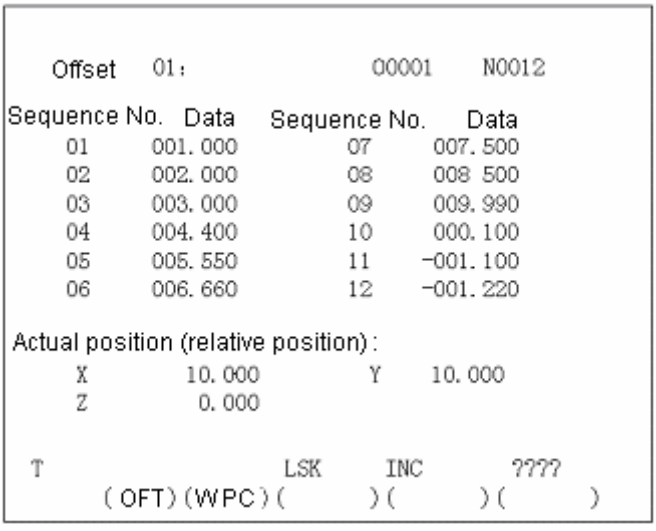
Position offset No. 37-48 (optional) of page 4;

⋮ ⋮

Position offset No. 97-99 or 97-108 (optional) of page 9;

⋮ ⋮

Position offset No. 193-200 (optional) of page 17;



The indication of page 1 of position offset

(3) Move the cursor to the offset number to be changed.

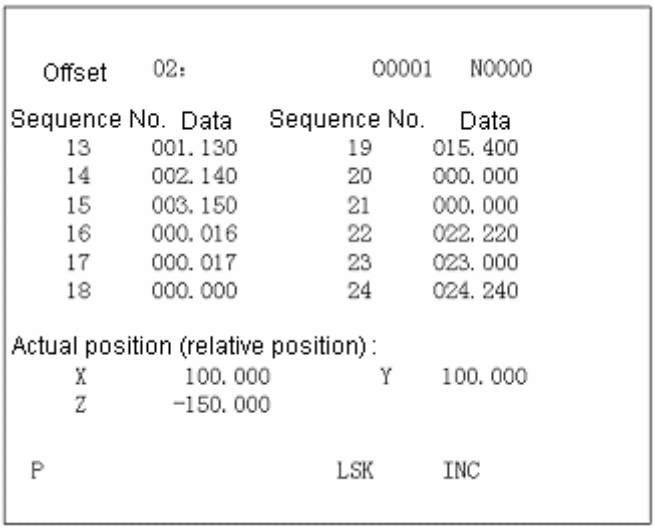
Method 1: Press the cursor button and move the cursor in succession. The next page will be switched to once the cursor goes beyond the current page.

Method 2: Set by typing with N, variable number and INPUT.

(4) Set the SELECTION MODE switch to a mode other than EDIT.

(5) Type [P] and [POSITION OFFSET] and then press the [INPUT] button.

The figure below is the page after [P], [1], [5], [•], [4] and [INPUT] is pressed when the position offset number is 19.

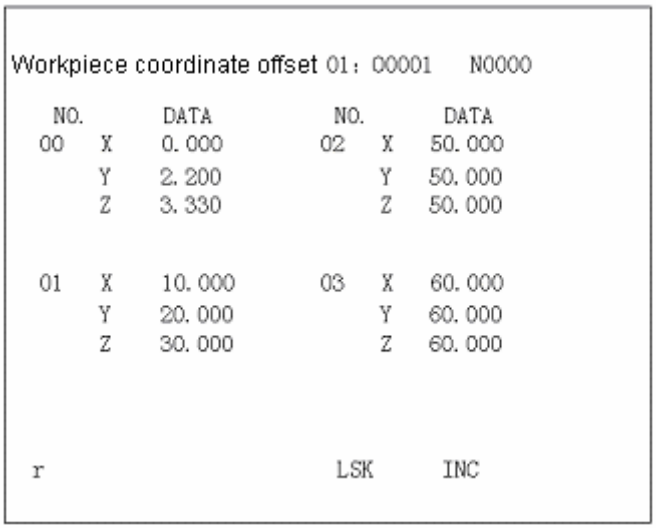


Note 1: When offset is changed in automatic operation, the new offset is not valid until its number is specified as D or H code.

Note 2: 0-9999 [INPUT] is used to reset all offsets to zero.

4.4.13 Setting and display of workpiece origin offset (Optional)

(1) Press [OFFSET] twice to display the WORKPIECE OFFSET page.



(2) Press the [PAGE] button to display the required page. Each page indicates as follows:

(i) Page 1(Workpiece coordinate offset 01)

00: Workpiece coordinate offset

01: The origin offset of the workpiece in workpiece coordinate system 1 (G54)

02: The origin offset of the workpiece in workpiece coordinate system 2 (G55)

03: The origin offset of the workpiece in workpiece coordinate system 3 (G56)

(ii) Page 2(Workpiece coordinate offset 02)

04: The origin offset of the workpiece in workpiece coordinate system 4 (G57)

05: The origin offset of the workpiece in workpiece coordinate system 5 (G58)

06: The origin offset of the workpiece in workpiece coordinate system 6 (G59)

(3) Move the cursor to the number to be changed.

Method 1: Press the cursor button [↑] or [↓] and move the cursor in succession. The next page will be switched to once the cursor goes beyond the current page.

Method 2: Set by typing with [N], [NUMBER] and [INPUT].

(4) Set the SELECTION MODE switch to a mode other than EDIT.

(5) Type [X], [Y], [Z] or [4TH/5TH] and the offset to be changed or set. Then press the [INPUT] button.

The setting range of the workpiece coordinates is 0 mm to ±7.999 mm or 0 inch to ±7.999 inch.

4.4.14 Measurement of tool length

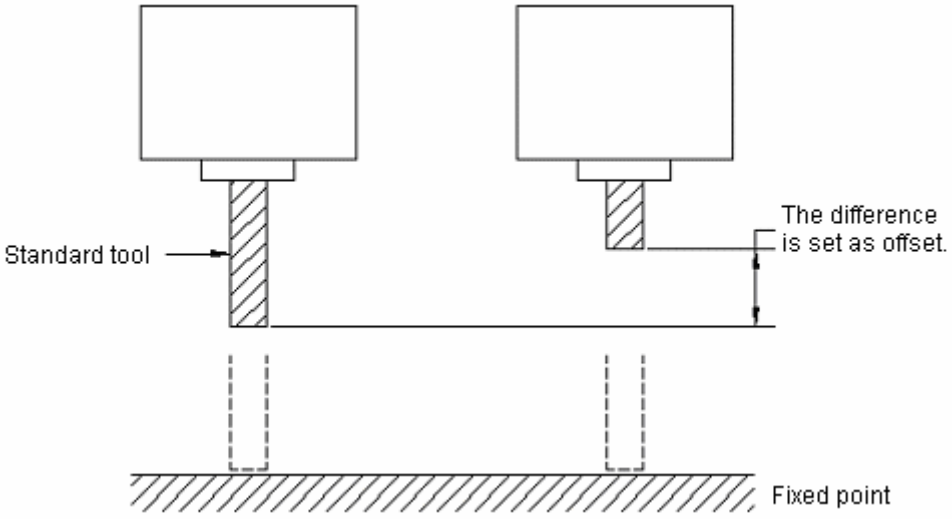
(1) Press the [OFFSET] button to select the page of offset.

(2) Select a standard tool and manually move it until it contacts the fixed point of the machine (or fixed point of workpiece).

(3) Press the [Z] and [SHIFT] buttons so that the relative coordinates of Axis Z is reset to zero.

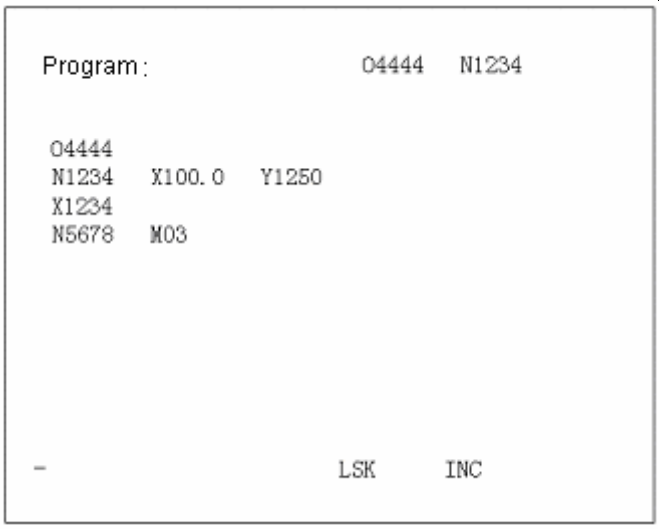
(4) Then select the tool to be measured and manually move it until it contacts the same fixed point. Now the difference between the standard tool and that to be measured is indicated in the display of relative position.

(5) Just like the setting of offset, move the cursor to the offset number and press the [Z] and [INPUT] keys but do not type in any numeral. The measured difference now is the offset to be input.



4.4.15 Program display (function button [PROGRAM])

(1) When in EDIT mode, press the [PROGRAM] key to display the page of the selected character in the current selection program.

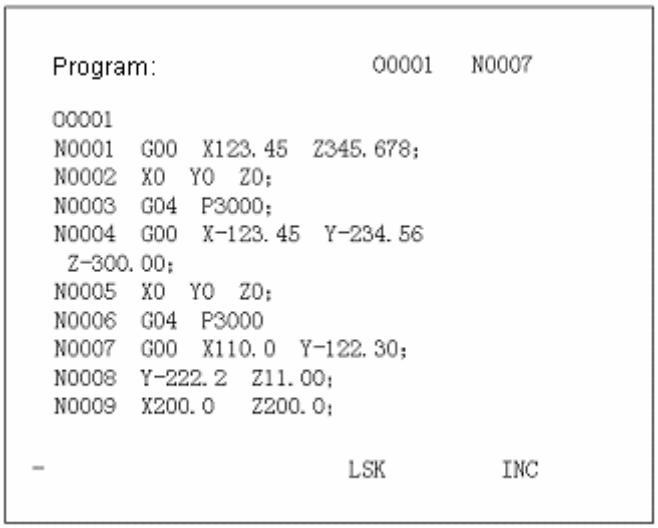


See the program number in Section 4.16 for which program is displayed. Pressing the cursor ⇓ or ↑ key displays the contents of the program in sequence. When the cursor ↓ key is pressed, the page is displayed in forward direction. When the cursor ↑ key is pressed, it is displayed in reverse direction.

(Note 1) Set the SELECTION MODE switch to EDIT and press the PROGRAM button to display the contents of the program at the beginning of an executing or executed block. However, the beginning of the program will be displayed when it is returned to (see 4.4.24.4).

(2) In automatic operation

Press the PROGRAM button to display the current executing block.



Indications of the cursor (in automatic operation)

(a) When the cursor blinks, it indicates the block to be executed next time.

(b) When the cursor does not blink, it indicates the currently executing or executed block.

(3) Note 1: Strictly speaking, when the buffer register becomes empty neither in automatic operating state nor in feed hold state, the blinking of the cursor indicates the next block is going to read in the buffer register so as to continue to execute a program.

Note 2: When the PAGE button or cursor button is pressed in EDIT mode to move the cursor and the program is started in memory mode, the block at the cursor in EDIT mode is read in the buffer register.

(4) EDIT mode and the other modes except automatic mode

When the PROGRAM button is pressed, the executing and executed blocks are displayed on the left side of the page and the blocks to be executed next time on the right side.

Note: When an angle moves in G28, G29, a fixed cycle and tool radius compensation, the contents on the left and right of the page are the same for the situation in which a block causes the cycle movement of several blocks.

4.4.16 Program number search (function key PROGRAM)

When several programs are stored in memory, it is possible to search one of them.

	0 1001	0 3054	0 1972

⟹ Search a program number

(1) Method 1

(a) Select a mode (EDIT or AUTO).

(b) Press the PROGRAM key.

(c) Enter O and the program number to be searched and then press the cursor ↓ key. The switching page of the program is displayed after search.

(2) Method 2

(a) Select the AUTO mode.

(b) Press the PROGRAM key.

(c) Press O, CANCEL and the cursor ↓ in sequence. The next stored program is displayed.

(3) Method 3

(a) Select the EDIT mode

(b) Press the PROGRAM key.

(c) Press O and the cursor ↓ to display the next stored program. In addition, the stored programs are displayed in sequence for reviewing the stored program numbers when the cursor ↓ key is pressed continuously.

Note 1: The start position is returned to when the stored program numbers are displayed.

Note 2: The contents in the buffer register are deleted when search a program number.

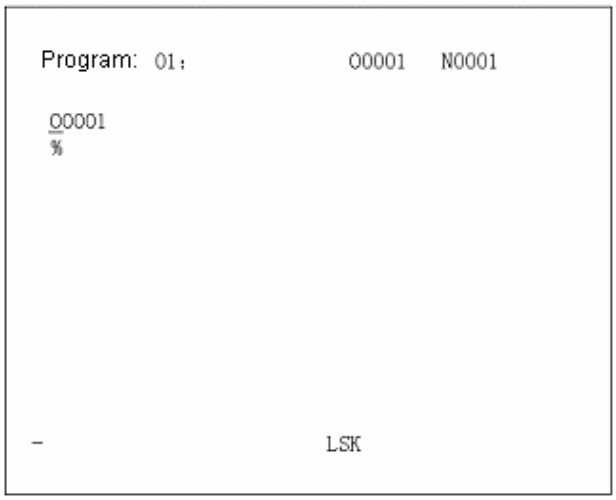
4.4.17 Inputting a program with keys

A program may be directly stored in memory with the **MDI** keys.

(a) Select the EDIT mode.

(b) Press the PROGRAM button to display the current program.

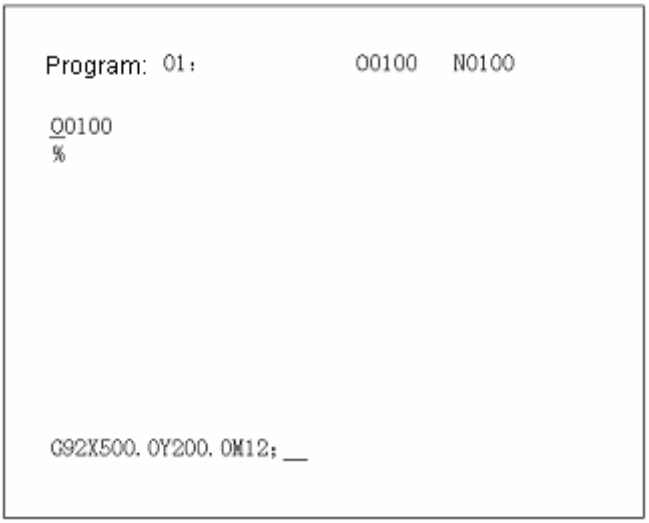
(c) Enter the program number to be stored. A new page appears when the O, the program number and INSERT keys are pressed.



(d) Type in a block

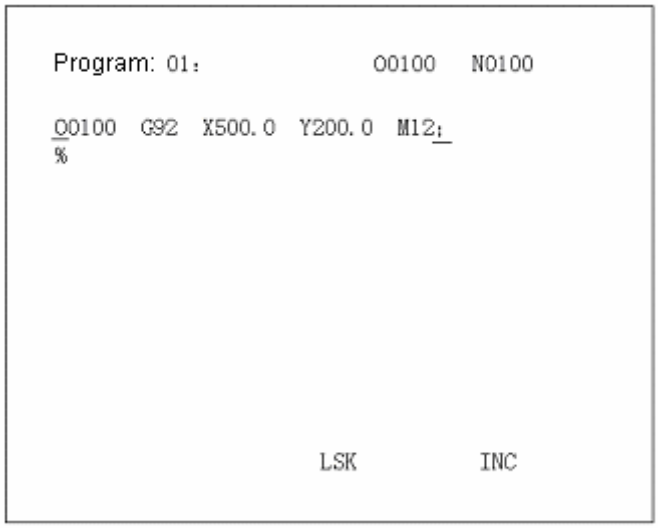
[Example] When typing in G92 X500.0 Y200.0 M12,

G	9	2	X	5	0	0	.	0	Y
2	0	0	.	0	M	1	2	EOB	



(e) If a typed character is incorrect, press the CANCEL key to delete the lastly typed word. Pressing the CANCEL key continuously deletes the typed words one by one from the last typed one. If the number of the characters of a block exceeds 32, the program cannot be entered. Now it is possible to divide the block with proper breakpoint.

(f) If the typed program is correct, press the INSERT key.



(g) Enter blocks in succession by this means.

(h) For correcting a typed block, proceed as indicated in the section of Program edit.

(i) For restart, continuously move the cursor to the lastly typed character. The procedure is the same as insertion.

(j) When all programs are input and at the end of the procedures, press the RESET key if you want to return to the start position.

4.4.18 Deletion of a program

(Program protection lock is active; function button PROGRAM) Deleting a program stored in memory:

(a) Select the EDIT mode.

(b) Press the PROGRAM button.

(c) Press O, the program number and DELETE. The program whose number is entered is deleted.

4.4.19 Deletion of all programs

(Program protection lock is active; function button PROGRAM) Deleting all programs stored in memory:

(a) Select the EDIT mode.

(b) Press the PROGRAM button.

(c) Press O, -, 9, 9, 9, 9 and DELETE.

4.4.20 Sequence number search

(Function button PROGRAM)

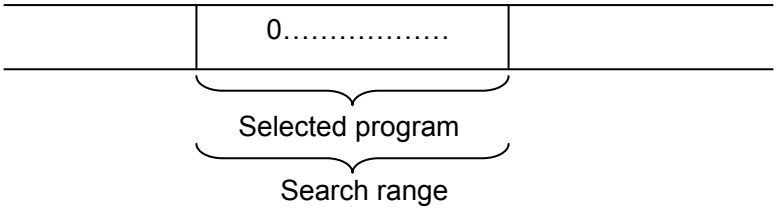
Sequence number search is usually used to search a sequence number in the midway of a program and start or restart the program from the block whose sequence number is searched. Its skipping over blocks exerts no influence on the NC system. Namely when skipping over blocks, the coordinates of the blocks who are skipped over, M, S, T or G codes do not change the coordinates and modal values of the NC. When a macro is supplied, sequence number will not be displayed in search.

Therefore, necessary M, S, T, G codes and coordinate system shall be set for the blocks to be started or restarted according to sequence number. If the block needs to restart search during machining, **MDI** must be used to assume M, S, T, G codes and coordinate system so that the

present state of the machine and NC system can be searched.

(a) Select the AUTO mode.

(b) Select the program number where the sequence number to be searched belongs to.



To search the sequence number on the block, follow (c). When the sequence number to be searched does not exist on the block, however, the program number with a pre-search sequence number shall be selected for sequence number search.

(c) Press the PROGRAM button.

(d) Type in N and the sequence number to be searched. Then press the cursor ↓ to find the sequence number.

Note 1: Coordinates and modal data do not update during search. These data are set through MDI after search.

Note 2: The following items are checked during search.

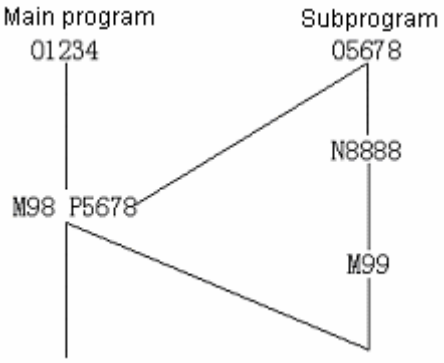
TH check

TV check

Skipping over optional blocks

Alarm check (03, 04, 05 and 10)

Note 3: M98P×××× (calling a subprogram) is not executed during sequence number search. Therefore, when the sequence numbers in the subprogram called by the current selected program for search in AUTO mode, No. 060 alarm will be given.



In the following example, alarm will be given when N8888 is searched.

4.4.21 Restart of a program

When the machine restarts after the damage of the tool and stop of machining, the restart function starts the machine from a block to be restarted according to the specified sequence number.

(1) The tool is damaged (method P)

(a) Press the FEED HOLD button, retract the tool and replace a new tool. Change the offset when necessary.

(b) Set the PROGRAM RESTART button on the operation panel to ON.

(c) Press the PROGRAM button to display the present program.

(d) Press the cursor ↑ button to return to the starting point of the program.

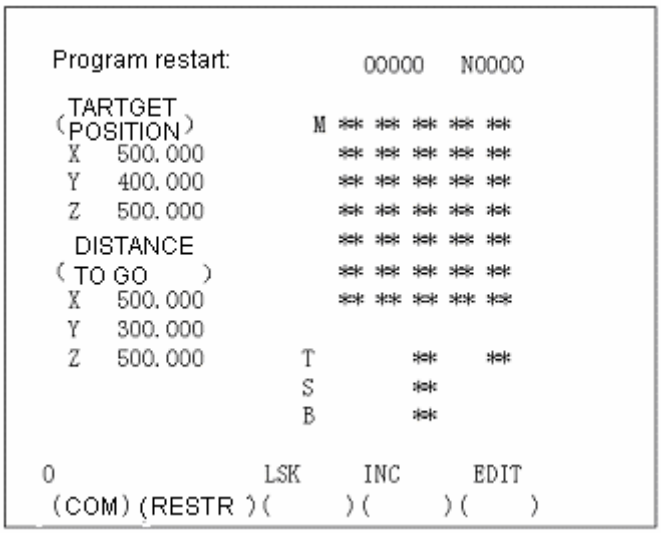
(e) Press the P, the sequence number and the cursor ↓ to search the block to be restarted. If the same sequence number appears for many times, e.g. when sequence number search calls a subprogram for many times, the higher four digits are specified as the number of times of block appearance and the lower four digits as its sequence number.

P 1 2 3 4 0 1 2 3 Cursor ↓

Number of times Sequence number

The number of times is 1, the higher four digits can be omitted. The preceding zero can also be omitted when the number of times is established.

(f) After search, LCD changes to display the page for program restart.



The TARGET POSITION indicates the restarting position of machining.

The DISTANCE TO GO indicates the distance from the current tool position to the restarting position of machining.

M indicates the M codes instructed in the last 35 times.

T indicates the T codes instructed in the last 2 times.

S indicates the last instructed S code.

B indicates the last instructed B codes.

The first instructed code is indicated.

The program restart instruction or the CYCLE START instruction for eliminating each code in reset state is also indicated.

(g) Set the program restart switch to OFF.

(h) Observe the page. Output by the MDI panel in MDI mode if the M, S, T and B codes to be output exist. In this case, the M, S, T and B codes to be output no longer appear on the program restart page.

(i) When the tool moves to the machining restarting position in AUTO mode, check the distance indicated by DISTANCE TO GO is correct and the tool does not contact workpiece. Press the CYCLE START button after manually moves the tool to the position that does not contact the workpiece. Now the tool moves to the restarting position in the sequence of the 4^{th} axis, axis X, axis Y and axis Z and restart machining.

(2) Restart machining (Q type) after the occurrence of the following conditions.

(a) Disconnect the power supply.

(b) Press the EMERGENCY STOP button

(c) The machine immediately stops due to storage type travel limit alarm.

(d) The coordinate system changes after the last automatic operation.

Example:

(i) Specify G92 instruction through **MDI**.

(ii) Move the coordinate system.

(iii) Set the automatic coordinate system after returning to the reference point.

(iv) Press the SHIFT button.

(v) Owing to reset, the change of coordinate system, etc.

(a) After switching on or the release of emergency stop and travel limit alarm, the machine returns to the reference point before restart (see the Notes below)

(b) The tool is moved to the programmed starting point of machining by manual and the set modal data and coordinate system is identical with the restart of the machine.

(c) If necessary, set or change the offset.

(d) Set the PROGRAM RESTART button on the operation panel of the machine to ON.

(e) Display the program by pressing the PROGRAM button. Search the required program when it is not available.

(f) Return the program to the origin point. Press the cursor ↓ button in automatic operating mode.

(g) Search the sequence number that the block restarts by pressing the Q and cursor ↓ button and typing in the sequence number.
When the same sequence number appears for many times during search, the higher four digits are specified as the number of times of sequence number appearance and the lower four digits as its sequence number.

(h) After search, LCD turns to display the page for program restart.

(i) Set the program restart switch to OFF.

(j) Observe the page. Output by the MDI panel in MDI mode if the M, S, T and B codes to be output exist. In this case, they no longer appear on the program restart page.

(k) When the tool moves to the machining restarting position, check that the tool does not contact workpiece. If necessary, manually move the tool to the position that does not contact the workpiece.

(l) Check that the distance indicated by the DISTANCE TO GO is adequate.

(m) Return to the AUTO mode and press the CYCLE START button. Now the tool moves to the restarting position in the sequence of the 4^{th} axis, axis X, axis Y and axis Z and restart machining.

Note 1: The following conditions, pressing the P button, the sequence number and the cursor ↓ button do not restart the program.

(a) After switching on, no automatic operation is performed.

(b) Automatic operation is performed after the release of emergency stop or storage type travel limit alarm.

(c) The coordinate system is established, changed or moved (the change in the offset of the origin of the external workpiece). The automatic operation is performed.

The above (a), (b) or 94—97 alarm reset causes P/S 97 alarm.

P/S 94 alarm caused by the establishment of a coordinate system;

P/S 95 alarm caused by the travel of a coordinate system;

P/S 96 alarm caused by the change of a coordinate system.

The block for the restartable machining is one of the many blocks. The block follows the block

when the coordinate system is last set of changed before the interruption of machinating.

Note 2: In P mode or Q mode, the tool moves to the machining restarting position by traveling by one axis each time. The stop of a single block is possible after the motion of the axis. However, JOG operation rather than MDI operation can be inserted. The returned axes cannot move.

Note 3: When move signal, offset and other conditions are different from the past, the tool cannot return to the machining restarting position identical with the past. The single block switch is set to ON or AUTO mode is switched to for continuous search operation.

Note 4: When feed hold is active during search or reset operation is performed after search, make sure to carry out program restarting operation from the beginning. After the end of search, however, the parameter No.007 “CLEAR Pos” shall be changed to reset state in MDI mode.
Note 5: CYCLE START can be ignored provided that the automatic restart switch of program is set to ON position.

Note 6: Always bring the JOG absolute switch for ON position for JOG operation no matter it is before or after machining.

When a program restarting operation instead of resetting is performed after JOG operation or when JOG operation is performed along the axis that has not returned to the machining restarting position, the concerned motion is performed once the manual absolute switch is set to ON position regardless that the manual absolute switch is in ON or OFF position.

Note 7: In principle, the tool cannot return to the correct position in the following cases:

(a) The manual absolute switch is brought to OFF position for JOG operation.

(b) The tool is moved in the lock state of the machine or after the cancellation of axis Z instruction.

(c) Mirror image function is used.

(d) The coordinate system is not set at the beginning of incremental Program edit.

(e) JOG operation is inserted during the return of an axis.

(f) Machine lock is disabled after the program restart is instructed.

(g) Program restart instruction is given during the execution of the cutting program with skip or the blocks of an absolute instruction.

(h) A coordinate system is established after search. Nevertheless, in the condition of (c) the return of the tool in P mode in the block executed by setting the mirror image machining switch to OFF position as well as the blocks that follow. In this case, the mirror image machining state is the same as interruption. No alarm is given in any case.

Note 8: When the specified block only includes M98, M99, macro calling instruction (M65, G66 and G67) or macro statement, or no specified block is searched, No.60 alarm will be given.

Note 9: When program restarting operation is instructed and G28 is detected without returning to the reference point after power on or cancellation of emergency stop or travel limit alarm (immediate stop), P/S alarm (98) will be resulted in.

Note 10: After the end of search, P/S alarm (99) is given when a move instruction is executed through MDI operations before axis motion.

Note 11: After the program restart is instructed, “RESTR blinks at the button of the LCD screen before the return of the last axis (Z).

Note 12: When the block before the restart block has G28, G30, an instruction or incremental instruction, the absolute position of the 4^{th} axis can be displayed in the range of 360°. In this case, the 4^{th} axis is an axis of rotation and the direction of returning to the reference point is negative.

4.4.22 Program number comparison stop function

The function is used to stop machining after an instruction is executed to the preset sequence number.

(a) Select the **MDI** mode.

(b) Press the SETTING button to display the SET page. Move the cursor to the set number 180 by pressing the cursor ↑ or ↓ key. The cursor cannot be moved with address N.

(c) Type the instruction in the sequence of P, the sequence number to be stopped and INPUT.

(d) Select the AUTO mode and get ready for operation by setting the machine.

(e) Press the CYCLE START button.

The machine stops after the data in the block whose sequence number has been predetermined in step (c).

The predetermined sequence number is cleared while the machine is stopping.

To perform another comparison stop, repeat the above procedures from (a).

Note 1: Sequence number O cannot be used for comparison stop

Note 2: The predetermined sequence number is cleared by setting.

4.4.23 Display of parameters (function button: PARAMETER)

Press the **PARAMETER** button to display the parameters, which are laid out in several pages. Find the desired parameter by pressing the PAGE button (see Appendix 5 for the meanings of parameters).

4.4.24 Program edit

(Function button: PROGRAM)

The function is used to modify the stored programs.

(1) Set the SELECTION MODE switch to EDIT.

(2) Press the PROGRAM button.

(3) Select a program. Proceed with (4) if the program has been selected; otherwise perform program number search.

(4) Search the words to be modified by scanning or word search.

(5) Modify, insert or delete the words.

Note 1: Definition and edit unit

A word comprises an address and the numeral that follows. For user macro, however, the concept of word is indefinite. Hence the concept of “edit unit” is adopted. Edit unit serves as the object of modification and deletion in a single operation and moves the cursor to its beginning in a single pass. For data insertion, the data is inserted behind the edit unit.

Definition of edit unit

① From one address to the next one;

② An address is a letter symbol: WHILE, GOTO, END, DO, =, or ; (EOB)

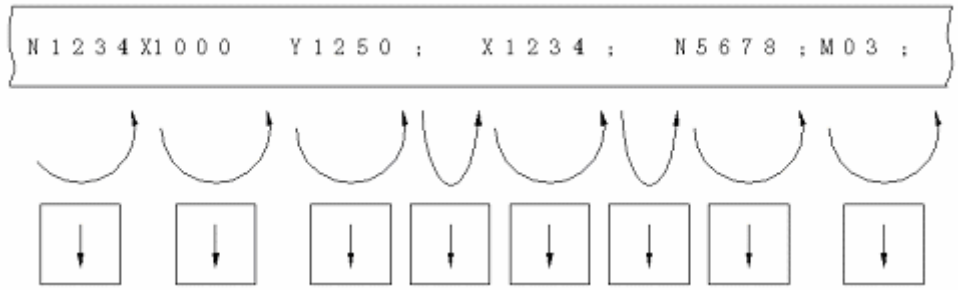
According to the definition, a word is also an edit unit. On the basis of the following explanations about edit, strictly speaking, word shall be called “edit unit”.

Note 2: During program execution, machining is temporarily suspended by single block skipping, feed hold and other functions. Nevertheless, continuing to execute a program is not allowed after program modification, insertion and deletion; otherwise the program cannot be correctly executed as required by program data. Program is displayed on the LCD after subsequent machining.

To modify stored data in part program edit mode, they must be modified in reset condition before program execution or when resetting operation is performed after edit.

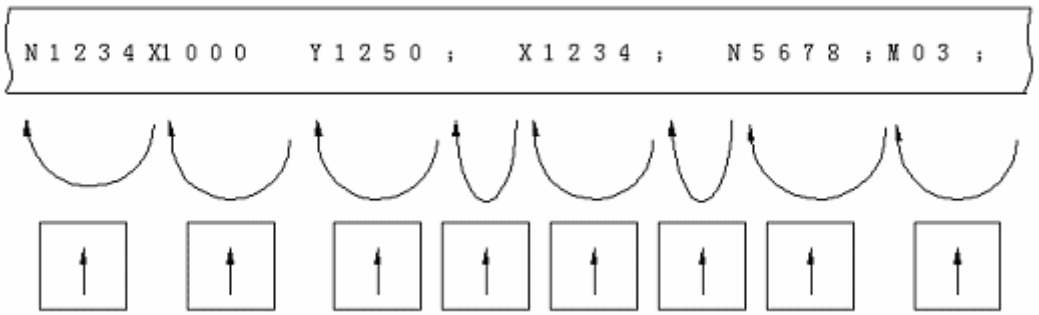
4.4.24.1 Word scanning

(1) Press the cursor [↓] button.



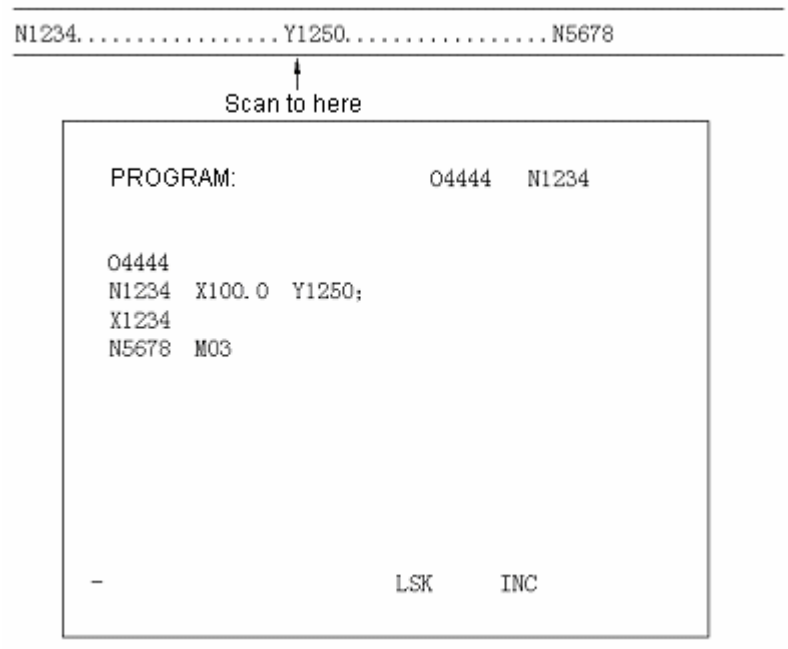
The cursor moves forward word by word on the screen. The cursor is displayed underneath the address character of the selected word.

(2) Press the cursor [↑] button.



The cursor returns word by word on the screen. The cursor is displayed underneath the address character of the selected word.

Example:



(3) Continuous search can be conducted by pressing down and holding the cursor [↓] or [↑] button.

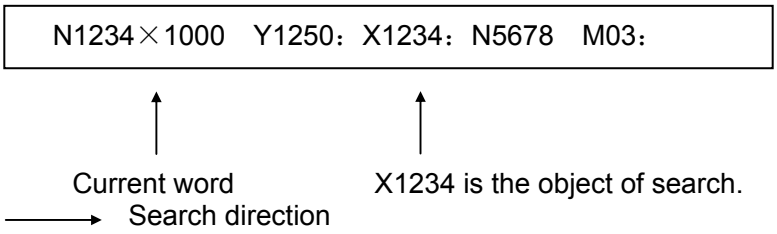
(4) The next page is displayed and search starts from the beginning of the page by pressing the page [↓] button.

(5) The previous page is displayed and search starts from the beginning of the page by pressing the page [↑] button.

(6) Displaying page by page is possible by pressing down and holding the cursor ↓ or ↑ button.

4.4.24.2 Word search

Search a specified word from the present position forward.



(1) Enter X, 1, 2, 3 and 4 with the keypad.

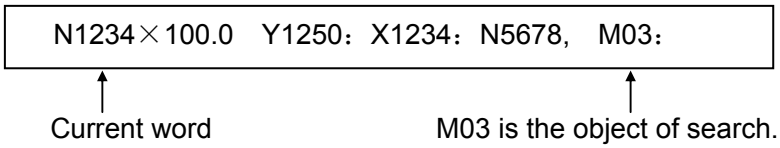
Note 1: The search that only enters X123 and X1234 with the keypad can not be performed.

Note 2: The search that only enters X9 and X009 with the keypad can not be performed. 009 shall be entered with the keypad for the search of X009.

(2) Search starts when the cursor ↓ is pressed. The cursor is displayed underneath X in X1234 after the search.

4.4.24.3 Address search

Search a specified address from the present position forward.



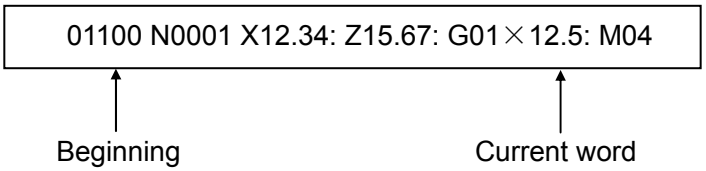
(1) Type in M.

(2) Search starts when the cursor ↓ is pressed. The cursor is displayed underneath M after the search.

Note 1: Pressing the CANCEL button after typing in a numeral deletes the latter and displays a blank space. Pressing the CANCEL button only indicates CAN.

Note 2: Both word search and address search do not start by pressing the cursor ↑ button.

4.4.24.4 Methods for returning to the beginning of a program



(1) Method 1

A program is displayed from its beginning when the RESET button is pressed in EDIT mode.

(2) Method 2

Perform sequence number search.

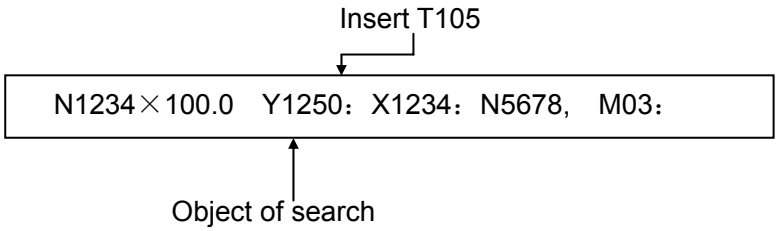
(3) Method 3

(a) Set the SELECTION MODE button to AUTO.

(b) Press the PROGRAM button.

(c) Press the cursor ↑ button to return to the EDIT mode for editing a part program.

4.4.24.5 Word insertion (active when the program protection lock is disabled)

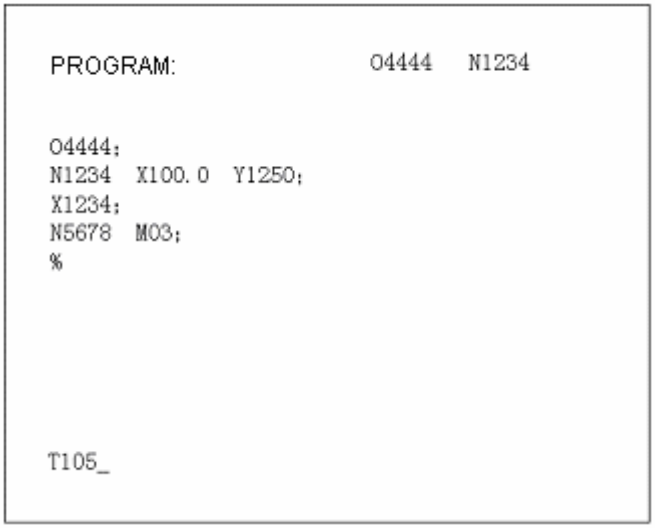


(1) Quickly search and scan the word preceding the position where a word is to be inserted.

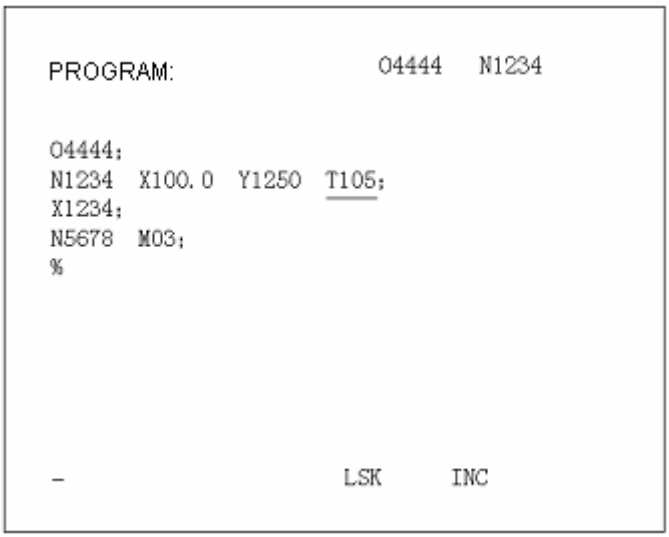
(a) Scan. See 4.24.1.

(b) See 4.24.2 for word search. When Y1250 is in front of the currently indicated position, first move the cursor to the beginning of the program.

(2) Type in T, 1, 0 and 5 and press the INSERT button.



Before insertion



After insertion

Note 1: When what is inserted is not an address but a numeral, the inserted number is added to the word indicated by the cursor. (Edit unit: In the above example, the insertion of 2.5 will create

Y12502.5 when the cursor is underneath Y in Y1250.

Note 2: A numeral can also be added to all addresses.

E.g. EOB, IF, etc. When the cursor is underneath ";", the insertion of 23 will create "; 23". However, it makes no sense in programming.

4.4.24.6 Word modification (active when the program protection lock is disabled)

```
N1234×100.0  Y1250：T105：  S1234：
```

↑

To be changed to M15

(1) Search and scan the word to be modified.

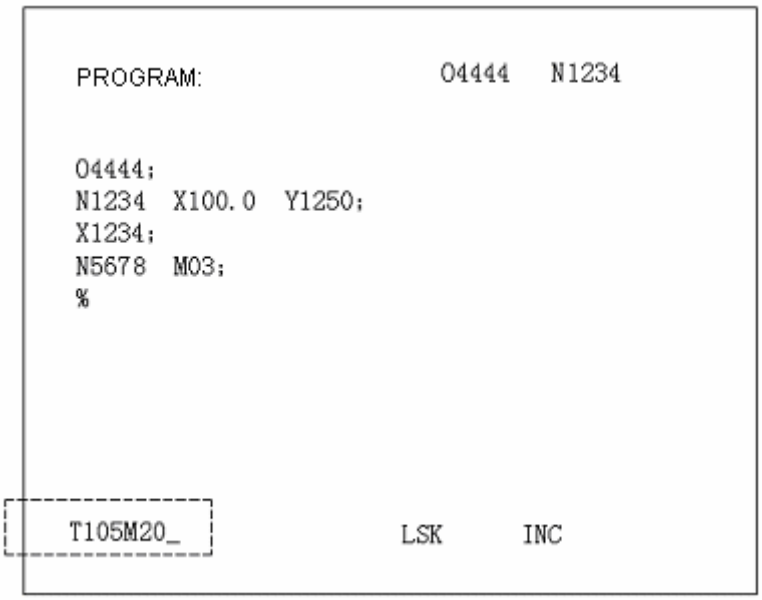
(2) Type in [M], [1] and [5] and press the [ALTER] button.

```
N1234×100.0  Y1250：M15：  S1234：
```

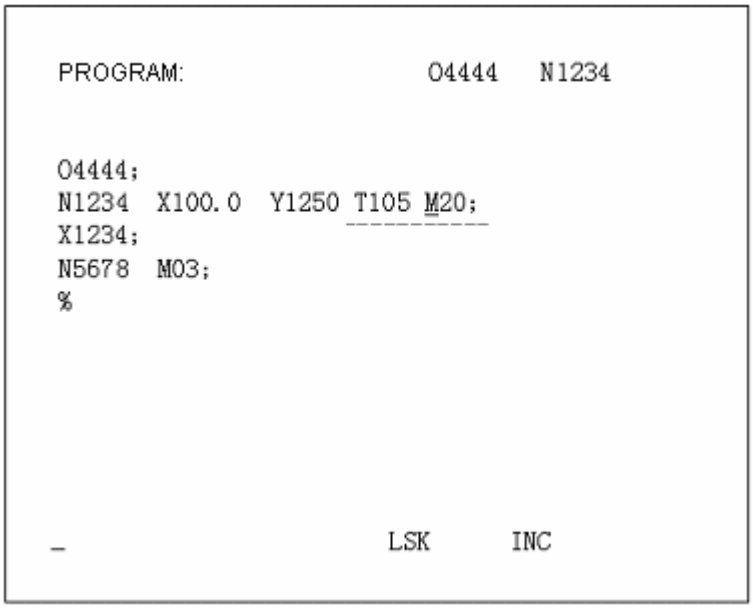
The program is to be modified.

4.4.24.7 Insertion and modification of several words, blocks and strings

Word, block, character string and other information can be inserted (up to 32 characters). In the example above, to insert T105 M20, type in T105 M20 and press the [INSERT] button.



Before insertion



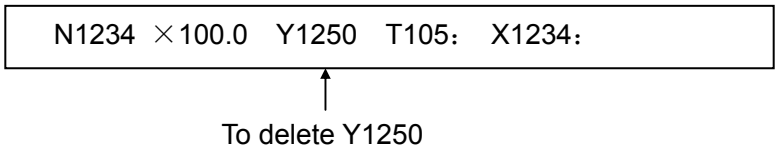
After insertion

A word indicated by the cursor can be changed to a word, block or a string, etc.

Note 1: When the cursor is underneath Y in Y1250, the insertion of 25 M20 will become Y1250 2.5 M20.

Note 2: When the cursor is underneath Y in Y1250 T105, the insertion of 2.5 M20 will become Y1250 2.5 M20.

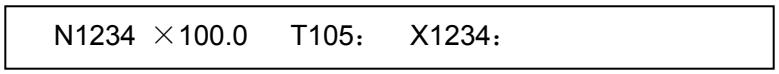
4.4.24.8 Word deletion (active when the program protection lock is disabled)



To delete Y1250

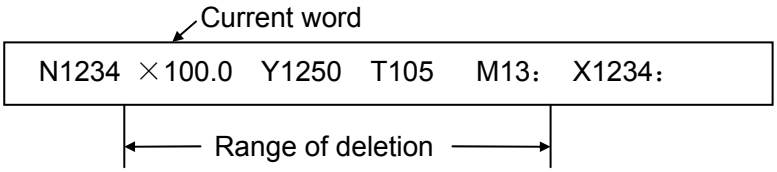
(1) Search and scan the word to be deleted.

(2) Press the DELETE button.



The program after deletion

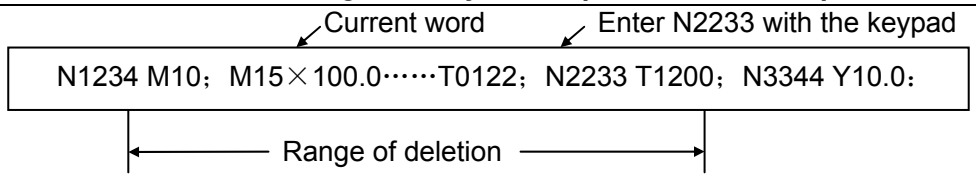
4.4.24.9 The deletion of the part before EOB



Press the EOB and DELETE buttons to delete the part before EOB and move the cursor to the underneath of the address character of the next word.

4.4.24.10 Deletion of blocks (active as program protection lock is disabled)

The range of deletion covers from the currently dedicated word to the block whose sequence number is specified.



(1) Type in the sequence number of the last block to be deleted. Type in [N], [2], [2], [3] and [3] in this example.

(2) Press the [DELETE] button.

4.4.24.11 Storage sorting

The frequent edit of part program sometimes prevents the storage from economic usage, resulting in the failure to store the program data whose length is specified. Hence it is necessary to regularly clean up the storage.

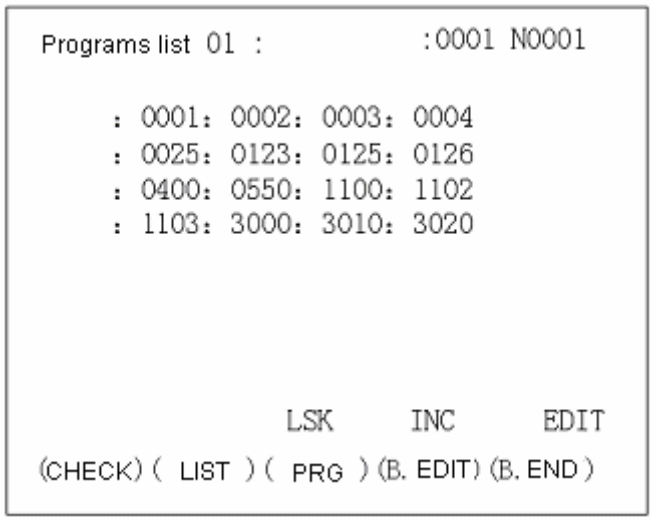
Press the [CANCEL] and [SHIFT] buttons in succession in the EDIT mode. After sorting, the admissible number of characters is indicated at the bottom of the screen.

Note 1: For one program, the storage stores the specified length of the program. For many programs, some storage areas are used to identify these programs.

Note 2: The storage areas to be modified or inserted that exceed the actual length are wasted in quick part program edit. Storage sorting may eliminate the waste.

4.4.24.12 The indication of all stored program numbers

Once the contents of the storage described in 4.24.11 are sorted, all stored program numbers are displayed.



4.4.24.13 Edit of user macro (active as the program protection lock is disabled)

User macro may be edited using the [SHIFT] key. Operate the key in EDIT mode and when the program protection switch is released. The following points shall be observed.

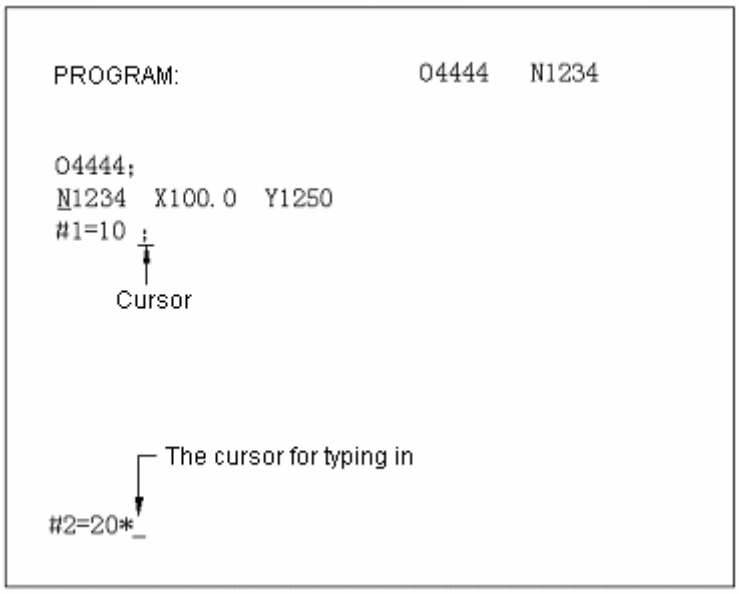
(a) [SHIFT] key

Once the [SHIFT] key is pressed, the cursor (the cursor for keyboard entry: the cursor is in the position of the character whose data is last input) changes from "—" to "∧". Press the key with a character in the lower right angle of the key top in this state. The character in the lower right angle can be entered.

After entering a character, the cursor restores to the initial "—". If the [SHIFT] key is pressed

twice, the cursor also restores the initial "—".

(Example)



(b) Deletion, insertion and modification of a program
While editing an entered user macro, the cursor moves at the following locations:

(i) At an address

(ii) At the / of an optional block skipped over

(iii) At the left beginning # of a substitution statement

(iv) At (·=)OR;

(v) At the leading character of IF, WHILE, GOTO, END or DO

On the LCD screen, there is a blank space for a character before the above characters. Deletion, modification and insertion may be performed between the front and rear cursor positions.

(Example) Cursor position

N001x-#100; #1=123; N002 / 2x[12/#3]:
N003x-SQRT[#3/3*[#4+1]]; N004x-# 2 Y#1;
N005#5=1+2-#10; IF[#1 NE 0]GOTO 10:
WHILE[#2LE·B]D01; #[2000+#2]=#2*10;
#2=#2+1: END1;

(Note 1) The cursor cannot be stopped in ( ).

Control OUT/IN

(Example)(#1=100):

The cursor cannot stop here.

(Note 2) The position of the cursor can change with the changed program.

(Example) X100 Y200; before program modification:

If Y200 is changed into 100 with the ALTER key, it will become X100 100;

(c) Abbreviations of macro word

To change or insert a macro word, it can be abbreviated to its first two characters. The underlined part can alternate the word as an abbreviation.

WHILE, GOTO, END, XOR, AND, SIN, COS, TAN,

ATAN, SQRT, ABS, BCD, BIX, FUP, ROUND.

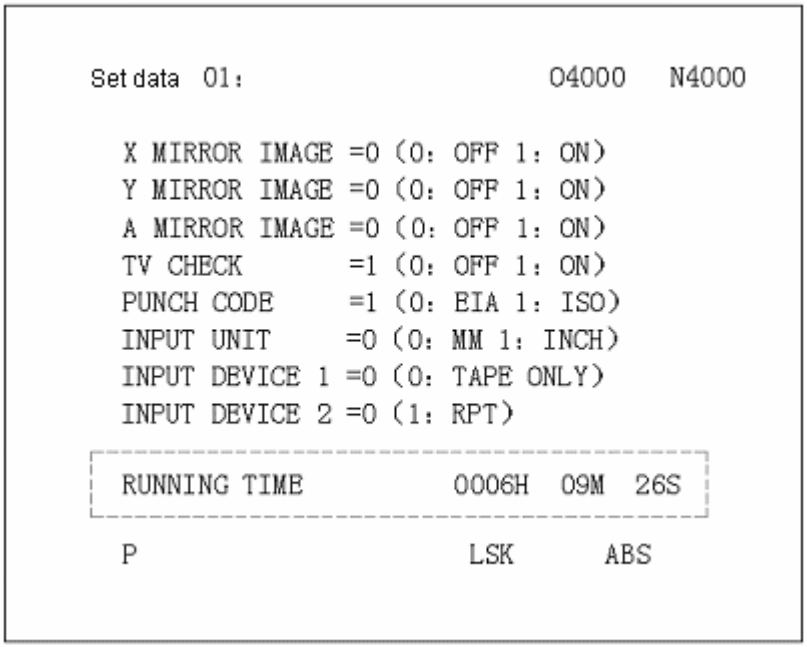
(Example): When WH[TA[#1*AB[#2]]LERO[#3]] is entered as data of keyboard entry, the actual entry will be:

WHILE[TAN[#1*ABS[#2]]LEROUND[#3]]

4.4.25 Indication of run time

Automatic operating time can be accumulated and displayed in hour, minute and second (increment of 2sec) on the screen.

Time is displayed as indicated in the picture below when the SETTING button is pressed. Press the PAGE button for other pages.


Set data 01:
O4000 N4000
X MIRROR IMAGE =0 (0: OFF 1: ON)
Y MIRROR IMAGE =0 (0: OFF 1: ON)
A MIRROR IMAGE =0 (0: OFF 1: ON)
TV CHECK =1 (0: OFF 1: ON)
PUNCH CODE =1 (0: BIA 1: ISO)
INPUT UNIT =0 (0: MM 1: INCH)
INPUT DEVICE 1 =0 (0: TAPE ONLY)
INPUT DEVICE 2 =0 (1: RPT)
RUNNING TIME 0006H 09M 26S
P
LSK
ABS

Note 1: The accumulated time includes automatic operating time but not the single block offtime, feed hold offtime, etc.

Note 2: If the power supply is switched off immediately after the stop of automatic operation, a time error up to 6min may be caused after power on again.

Note 3: When necessary, time can be preset through setting operations. The data number is 57, 58 or 59.

4.4.26 Menu switching function

It is possible to enable or disable the switching function of the NC storage using the LCD instead of the operation panel of the machine.

With this function, the number of switches on the operation panel can be reduced. The following signals may be switched on or off through the LCD.

(1) Single block (SBK)

(2) Machine lock (MLK)

(3) Dry run (DRN)

(4) Optional block skip 1 to 9(BDT1—9)

(5) Mirror image (MIX, MIY, M14 and M15)

(6) Display lock (DLK)

(7) Auxiliary function lock (AFL)

(8) Axis Z neglected (ZNG)

(9) JOG absolute (ABS)

After the signal data is stored in memory, the switching of these data displayed on the LCD remains unchanged even the NC power is disconnected.

These signals are not completely replaced by those displayed on the LCD. These signals are deemed to be switched on no matter whether the machine signals or the signals set on the LCD are switched on. Accordingly, these signals can be switched on or off through the LCD by disconnecting the above listed optional signals through the operation panel.

Setting and display: The LCD state of the above signals may be displayed through the following operations.

Display

(i) Select the set chapter 3

For setting display, press the SETTING button and then press it twice again.

(ii) Press the PAGE button to select the desired page from the two pages than can be displayed.

Page 1: The part other than optional blocks skipping

Page 2: Optional block skipping 1~9.

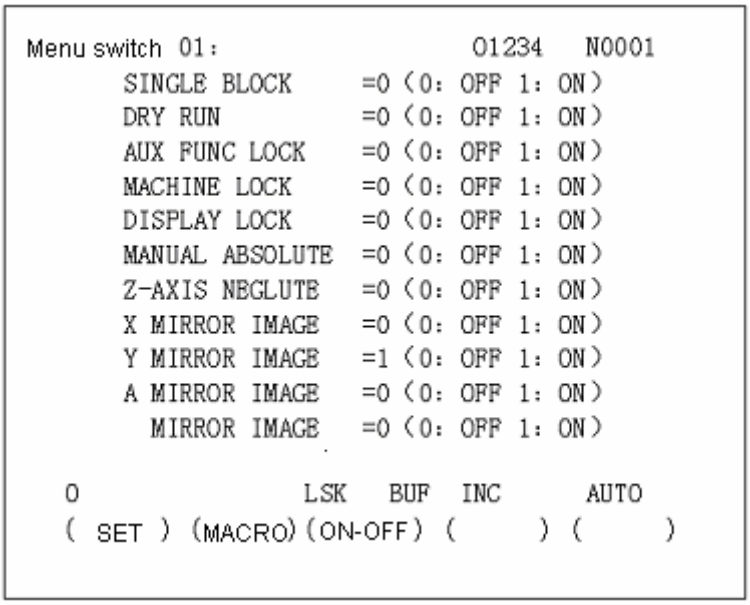
Setting

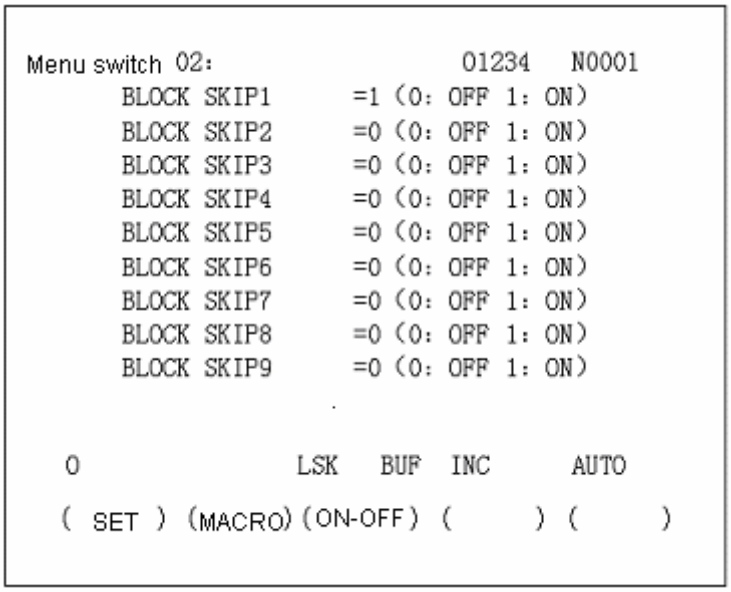
Proceed as follows after the above procedures.

(iii) Move the cursor to the page number to be changed.

Move the cursor to the page number to be changed by pressing the cursor ↑ or ↓ key.

(iv) When the address P is pressed, enter 1 for switching on and 0 for switching off. Press the keys in the sequence of (P) {1 2} (entry).


Menu switch 01: O1234 N0001
SINGLE BLOCK =0 (0: OFF 1: ON)
DRY RUN =0 (0: OFF 1: ON)
AUX FUNC LOCK =0 (0: OFF 1: ON)
MACHINE LOCK =0 (0: OFF 1: ON)
DISPLAY LOCK =0 (0: OFF 1: ON)
MANUAL ABSOLUTE =0 (0: OFF 1: ON)
Z-AXIS NBGLUTE =0 (0: OFF 1: ON)
X MIRROR IMAGE =0 (0: OFF 1: ON)
Y MIRROR IMAGE =1 (0: OFF 1: ON)
A MIRROR IMAGE =0 (0: OFF 1: ON)
MIRROR IMAGE =0 (0: OFF 1: ON)
0 LSK BUF INC AUTO
(SET) (MACRO) (ON-OFF) ( ) ( )


Menu switch 02:
01234 N0001
BLOCK SKIP1 =1 (0: OFF 1: ON)
BLOCK SKIP2 =0 (0: OFF 1: ON)
BLOCK SKIP3 =0 (0: OFF 1: ON)
BLOCK SKIP4 =0 (0: OFF 1: ON)
BLOCK SKIP5 =0 (0: OFF 1: ON)
BLOCK SKIP6 =0 (0: OFF 1: ON)
BLOCK SKIP7 =0 (0: OFF 1: ON)
BLOCK SKIP8 =0 (0: OFF 1: ON)
BLOCK SKIP9 =0 (0: OFF 1: ON)
0 LSK BUF INC AUTO
(SET) (MACRO) (ON-OFF) ( ) ( )

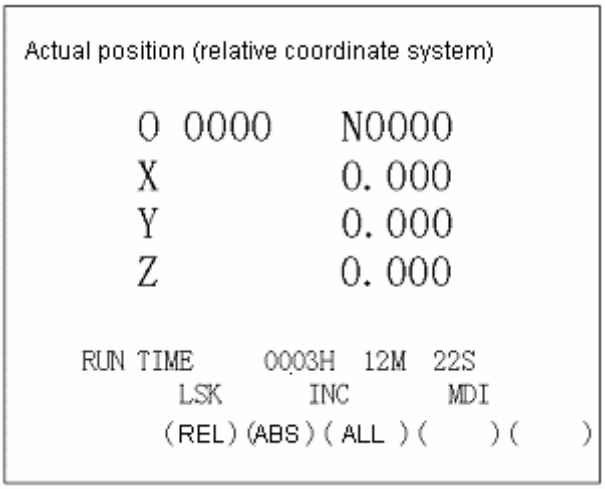
4.4.27 Operations of LCD soft function keys

4.4.27.1 General

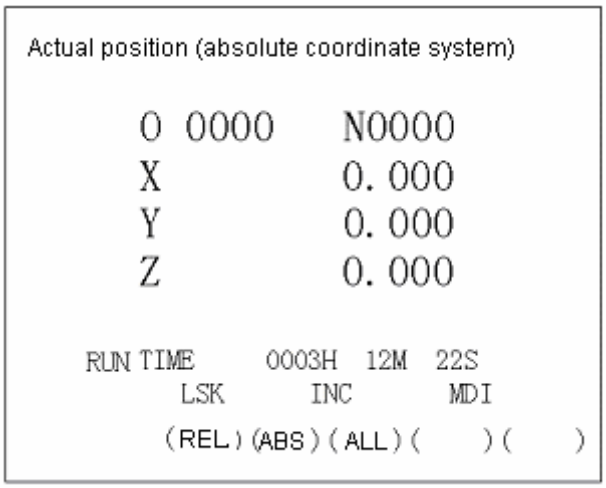
Here function buttons (POSITION, PROGRAM, OFFSET etc) are used as soft function keys. Their meanings may be displayed on the LCD. All the pages abstained by pressing the soft function keys are described below.

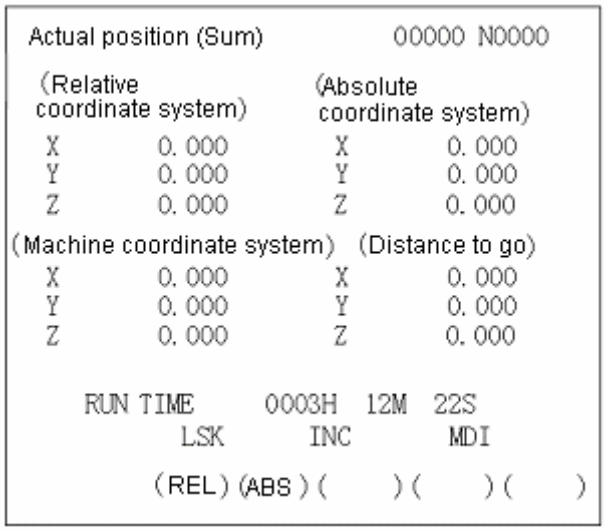
4.4.27.2 Display

(1) Display of actual position



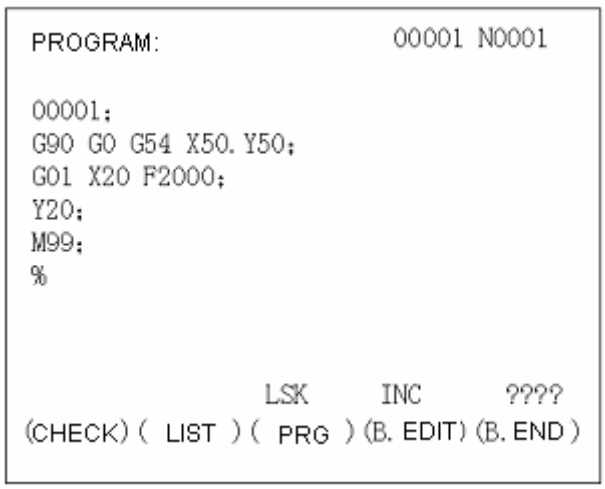
The actual position of relative coordinate system is displayed on the LCD when the soft function key REL is pressed.





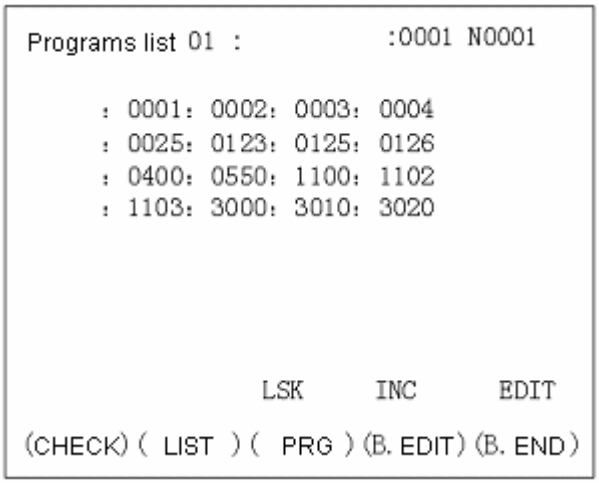
Press the page ↓ key the update display as shown above. The positions of workpiece coordinate system and relative coordinate system are exchangeable on the screen.

(2) Display of a program

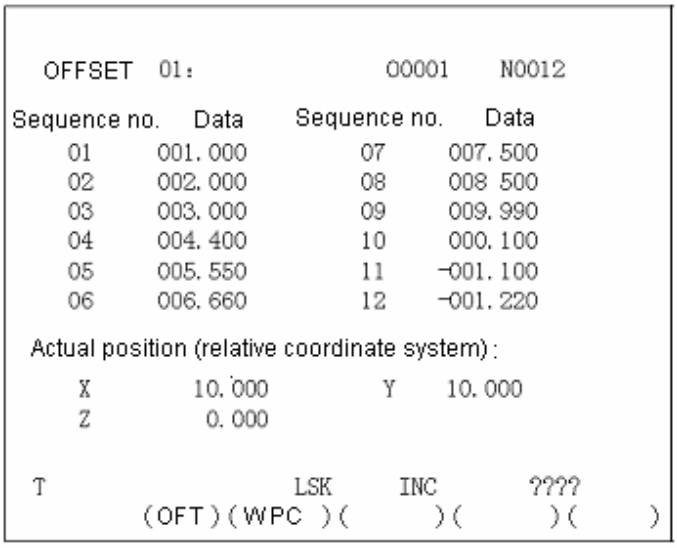


Press the PROGRAM soft function key to display a program. It is possible to make background edit (edit the programs other than the currently running programs) using B. EDIT.

Press the PROGRAM soft function key again to display program list. The program list is displayed as follows.

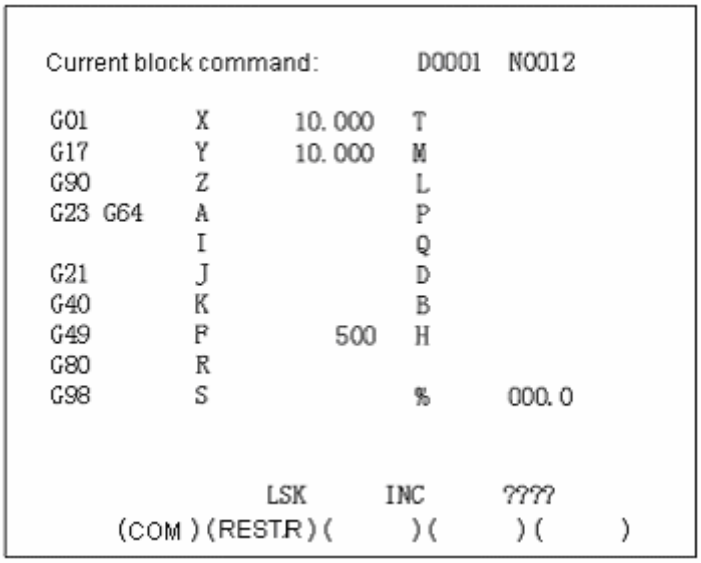


(3) Display of offsets

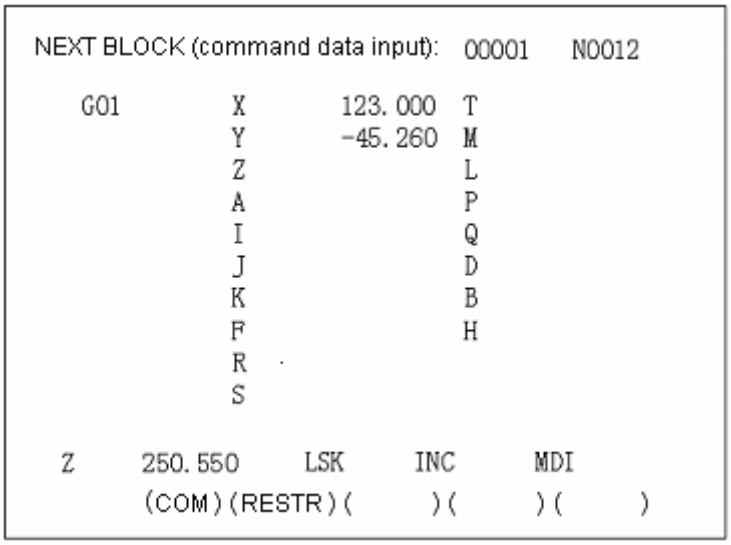


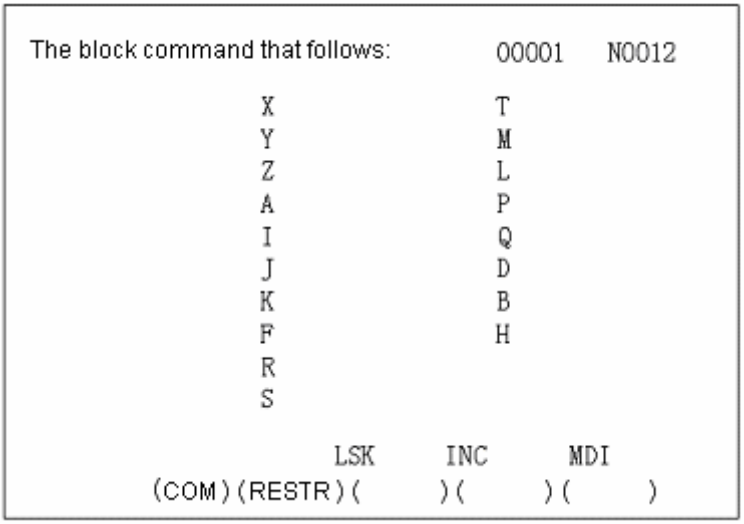
When the OFT soft function key is pressed, the offset corresponding to each tool number is indicated on the LCD. Press the page ↑ or ↓ key and cursor ↑ or ↓ key to select the desired data. At the same time, the actual position of relative coordinate system is displayed on the lower part of the LCD.

(4) Display of an instruction

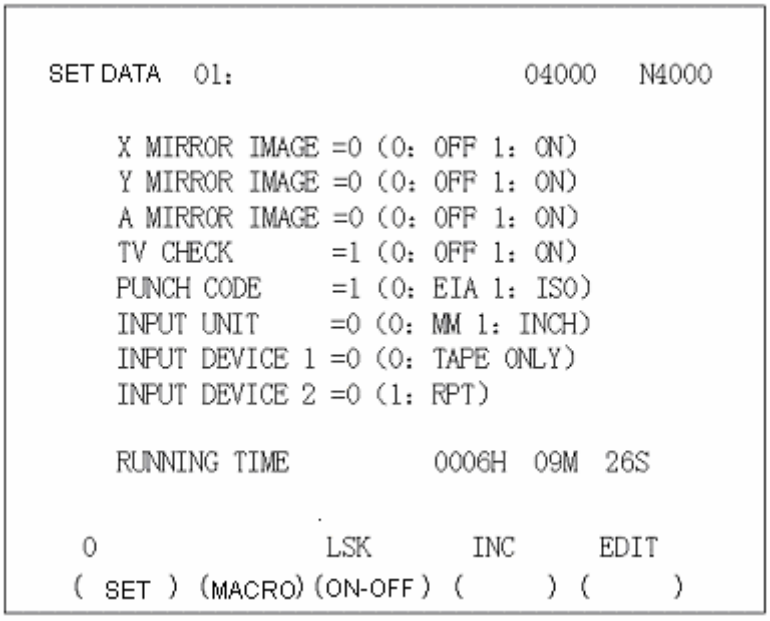


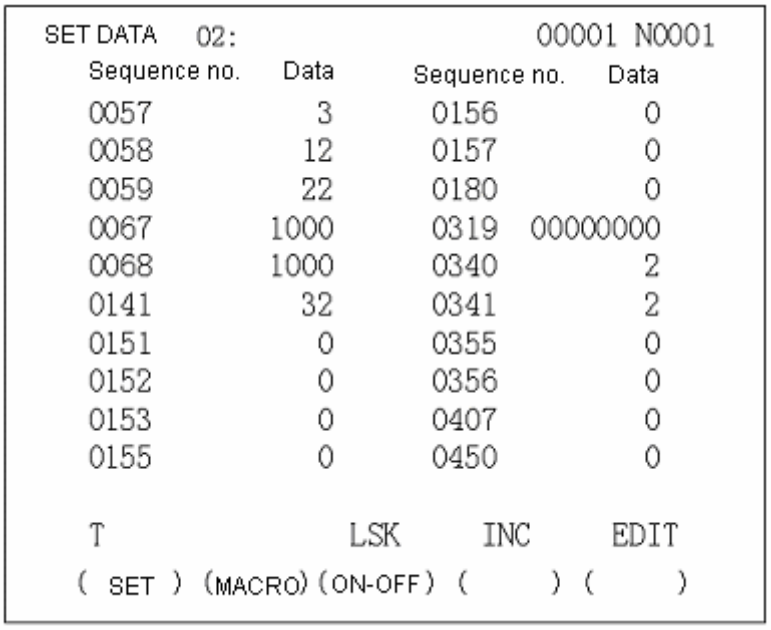
Press the COMMAND soft key. The data displayed on three pages of the LCD appears on the screen as shown in the above figure. The screen changes as follows when the page ↓ key is pressed.



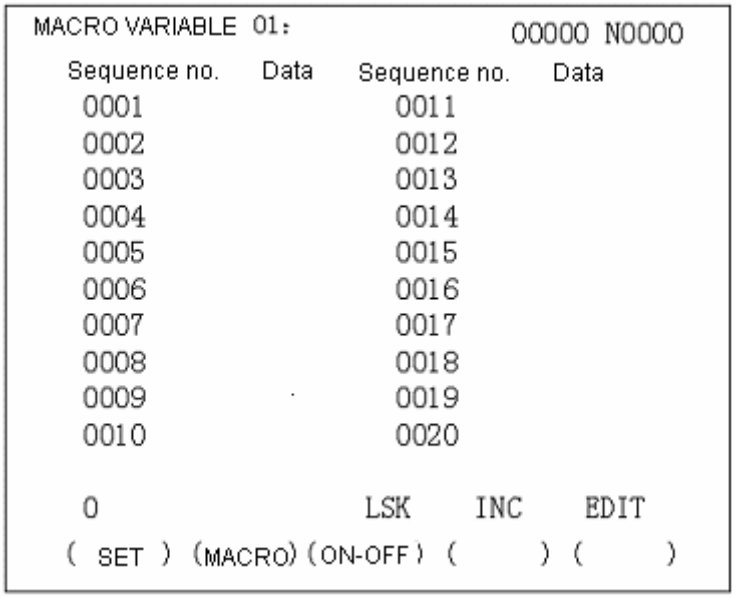


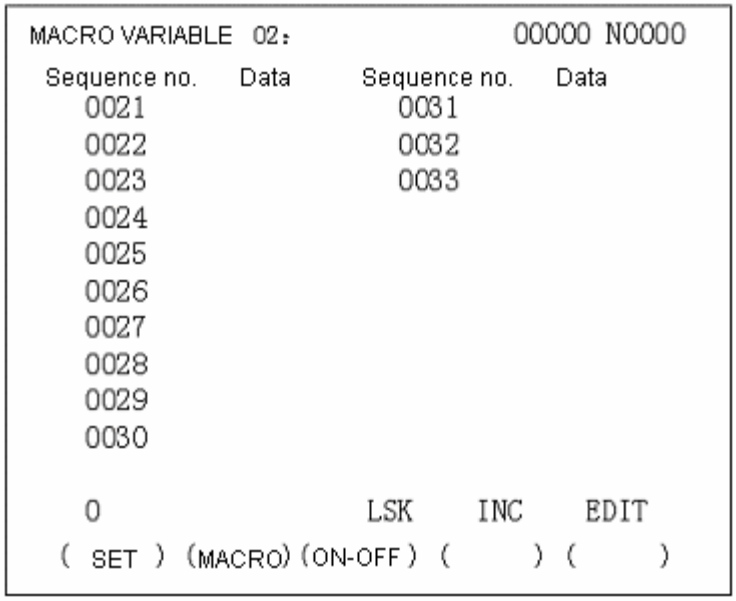
(5) Setting


SET DATA 01: 04000 N4000
X MIRROR IMAGE =0 (0: OFF 1: ON)
Y MIRROR IMAGE =0 (0: OFF 1: ON)
A MIRROR IMAGE =0 (0: OFF 1: ON)
TV CHECK =1 (0: OFF 1: ON)
PUNCH CODE =1 (0: EIA 1: ISO)
INPUT UNIT =0 (0: MM 1: INCH)
INPUT DEVICE 1 =0 (0: TAPE ONLY)
INPUT DEVICE 2 =0 (1: RPT)
RUNNING TIME 0006H 09M 26S
0 LSK INC EDIT
(SET) (MACRO) (ON-OFF) ( ) ( )



Press the SET soft function key. Now all set data is displayed on the screen. Run time is displayed in the page of SET DATA 01.

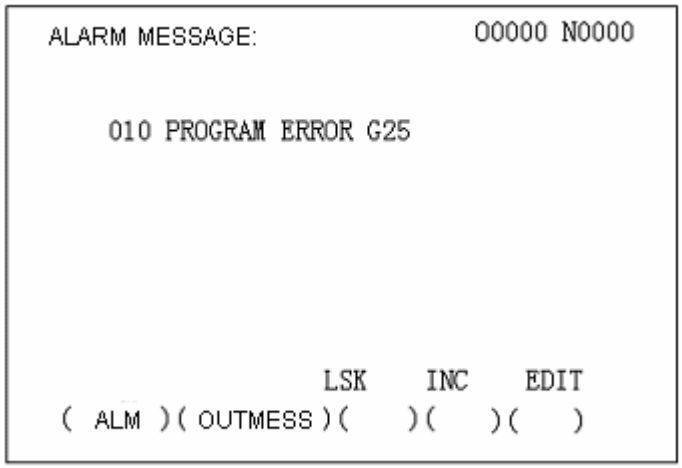




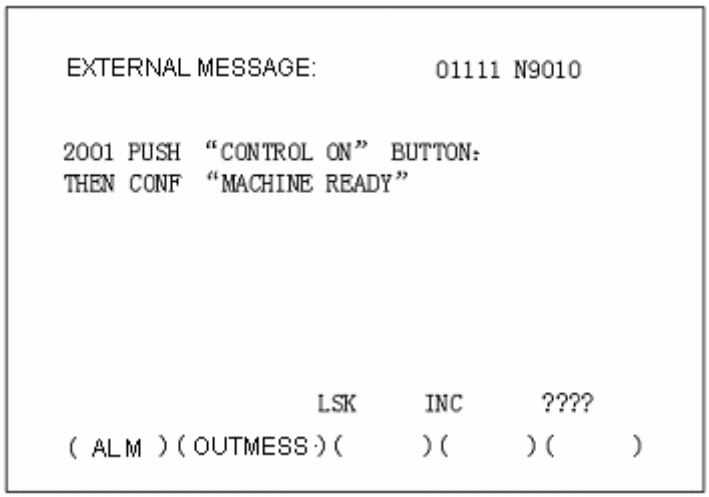
Press the MACRO key. Now the local variable and common variable of user macro are displayed on the LCD screen.

Press the ON-OFF key. The menu switch window is displayed as shown as shown in Section 4.26 (menu switch function).

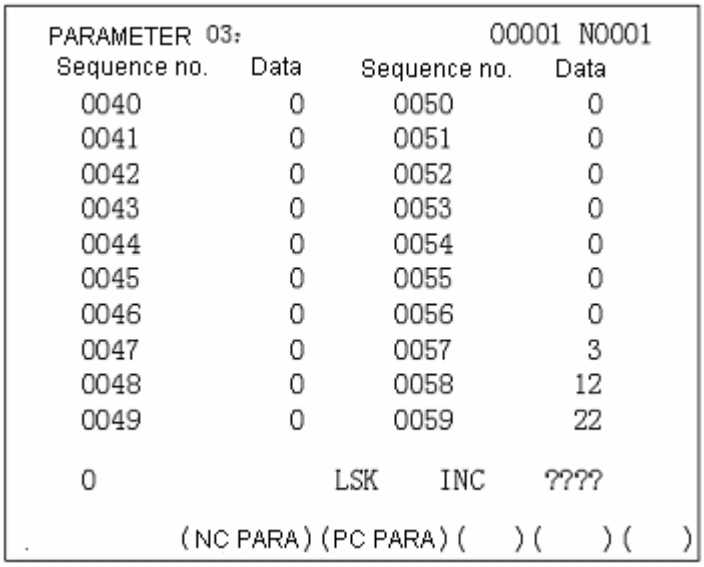
(6) Display of alarm and operation information



Pressing the ALM soft function key displays the alarm messages as shown in the above figure. Pressing the OUTMESS soft function key displays the external information as shown in the above figure.



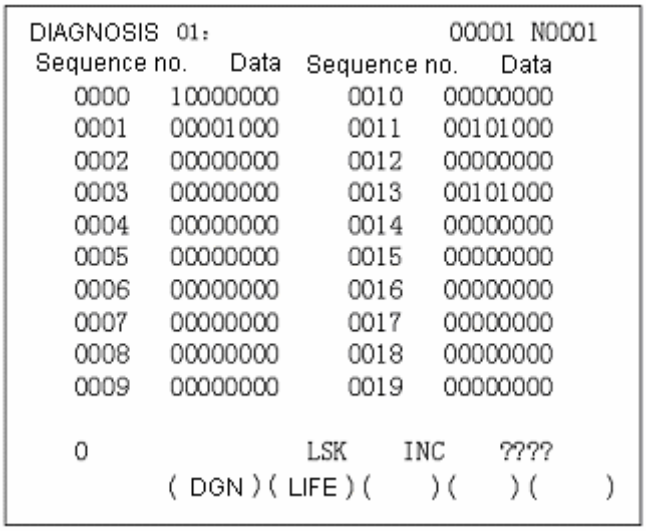
(7) Parameter display



Pressing the NC PARA and PC PARA soft function keys to display the relevant parameters on the LCD.

Pressing the page ↑ or ↓ and cursor ↑ or ↓ to display the desired parameters.

(8) Display of diagnostic data



Pressing the DGN soft function key displays 64 diagnostic data.

Pressing the page ↑ or ↓ and cursor ↑ or ↓ to display the desired diagnostic data.

4.4.27.3 Direct entry of measured workpiece origin offset

The coordinates of the relative coordinate system that offsets at the workpiece origin and is indicated by the LCD screen may be set as workpiece origin offset. With the function the relative coordinate system is cleared away at the reference point and the machine is moved to the workpiece origin by JOG. Now the coordinates of the relative coordinate system may be set as workpiece origin offset. In this way it is easy to set the workpiece origin offset.

(1) Operating procedures

The relative coordinate system may be cleared away and workpiece origin offset set on the workpiece origin offset page of the LCD through the following procedures.

(a) Delete the relative coordinate system

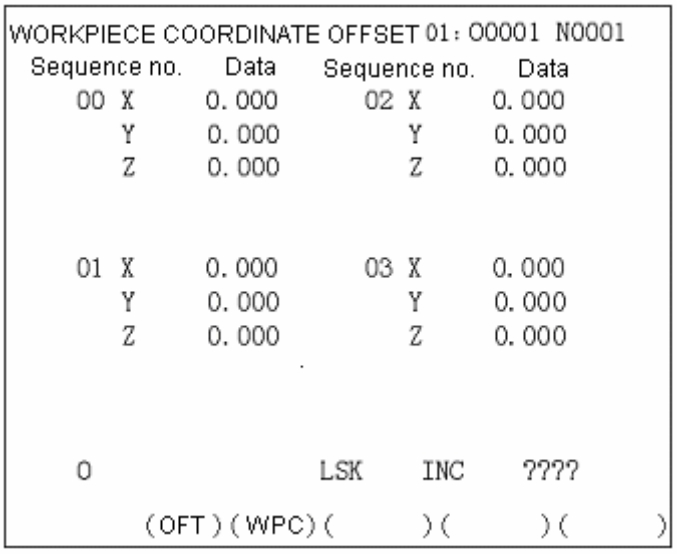
Pressing the X and SHIFT keys deletes the relative coordinate system of axis X (this operation accordingly applies to axes Y and Z as well as the 4th and 5th axes).

(b) Set workpiece origin offset

Press the X and INPUT keys after moving the cursor to the required workpiece offset number. Then the coordinates of the axis X in the corresponding coordinate system is set as the axis-X workpiece origin offset of the selected workpiece offset number.

(This operation accordingly applies to axes Y and Z as well as the 4th and 5th axes

Note: When the X key is pressed in steps (a) and (b), the X of the relative coordinate system blinks.


WORKPIECE COORDINATE OFFSET 01: O0001 N0001
Sequence no. Data Sequence no. Data
00 X 0.000 02 X 0.000
Y 0.000 Y 0.000
Z 0.000 Z 0.000
01 X 0.000 03 X 0.000
Y 0.000 Y 0.000
Z 0.000 Z 0.000
0 LSK INC ????
(OFT) (WPC) ( ) ( ) ( )

4.5 Position Indication through Position Display Unit (Available upon Customer's Request)

The position display unit indicates the actual position. The position display reset buttons are connected to the position display unit. Each button corresponds to one axis. The position display of the relevant axis is cleared when any one of the buttons is pressed.

After the absolute origin (G92) is programmed and established, it is indicated on the display unit through parameter setting (PPD) coordinates.

Note 1: Position pulse cannot be sent to the position display unit when the DISPLAY LOCK switch is set to ON position. The switch may used to prevent writing offset in position indication by manually moving the coordinate system.

Note 2: Position is indicated in Inch-system format in the event of entry in inch system.

Position is indicated in metric format in the event of entry in metric system. The RESET button shall be pressed so that the position indication is zero when the input system is switched from inch system to metric system, vice versa.

Note 3: Machine compensation data, e.g. amount of clearance compensation cannot be indicated on the position display unit.

Appendix 1 Codes for Programming

ISO Code										EIN Code										Meaning	
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		
0			○	○		0				**0**			○			○					Numeral “0”
1	○		○	○		0			○	**1**						○			○		"1
2	○		○	○		○		○		**2**						○		○			"2
3			○	○		○		○	○	**3**				○		○	○	○	○		"3
4	○		○	○		○	○			**4**						○	○				"4
5			○	○		○	○		○	**5**				○		○	○		○		"5
6			○	○		○	○	○		**6**				○		○	○	○			"6
7	○		○	○		○	○	○	○	**7**						○	○	○	○		"7
8	○		○	○	○	○				**8**					○	○					"8
9			○	○	○	○			○	**9**				○	○	○			○		"9
A		○				○			○	**a**		○	○			○			○		Address A
B		○				○		○		**b**		○	○			○		○		?	"B
C	○	○				○		○	○	**c**		○	○	○		○		○	○	?	"C
D		○				○	○			**d**		○	○			○	○				"D
E	○	○				○	○		○	**e**		○	○	○		○	○		○		"E
F	○	○				○	○	○		**f**		○	○	○		○	○	○			"F
G		○				○	○	○	○	**g**		○	○			○	○	○	○		"G
H		○			○	○				**h**		○	○		○	○				?	"H
I	○	○			○	○			○	**i**		○	○	○	○	○			○		"I
J	○	○			○	○		○		**j**		○		○		○			○	?	"J
K		○			○	○		○	○	**k**		○		○		○		○			"K
L	○	○			○	○	○			**l**		○				○		○	○		"L
M		○			○	○	○		○	**m**		○		○		○	○				"M
N		○			○	○	○	○		**n**		○				○	○		○		"N
O	○	○			○	○	○	○	○	**o**		○				○	○	○			"O
P		○		○		○				**p**		○		○		○	○	○	○		"P
Q	○	○		○		○			○	**q**		○		○	○	○					"Q
R	○	○		○		○		○		**r**		○			○	○			○		"R
S		○		○		○		○	○	**s**			○	○		○		○			"S
T	○	○		○		○	○			**t**			○			○		○	○		"T
U		○		○		○	○		○	**u**			○	○		○	○				"U
V		○		○		○	○	○		**v**			○			○	○		○	?	"V
W	○	○		○		○	○	○	○	**w**			○			○	○	○			"W
X	○	○		○	○	○				**x**			○	○		○	○	○	○		"X
Y		○		○	○	○			○	**y**			○	○	○	○				?	"Y
Z		○		○	○	○		○		**z**			○		○	○			○		"Z
DEL	○	○	○	○	○	○	○	○	○	**DEL**		○	○	○	○	○	○	○	○	*	Delete (cancellation of an incorrect hole)

ISO Code										EIN Code									Meaning	
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		
NUL						0				**Blank**						0				*	No-hole and EIA codes are not used in effective-value information area
BS	○				○	0				**BS**			○		○	0		○		*	Rewind (backspace)
HT					○	0			○	**Tab**			○	○	○	0	○	○		*	Separator
LForNL					○	0		○		**CRorEOB**	○					0					End of block
CR	○				○	0	○		○							0				*	Return of printer base
SP	○		○			0				**SP**				○		0				*	Blank space
%	○		○			0	○		○	**ER**					○	0		○	○		End of rewinding
(			○		○	0				**(2-4-3)**				○	○	0		○			Control OUT (start of note)
)	○		○		○	0			○	**(2-4-7)**		○			○	0		○			Control IN (end of note)
+			○		○	0		○	○	**+**		○	○	○		0				*	Positive sign
—			○		○	0	○		○	—		○				0					Negative sign
:			○	○	○	0		○								0					As a program number of ISO codes
/	○		○		○	0	○	○	○	**/**			○	○		0			○		Stop of block selection
.			○		○	0	○	○		**.**		○	○		○	0		○	○		Full stop (decimal point)
#	○		○			0		○	○							0				*	Well number
$			○			0	○									0				*	Unit number
&	○		○			0	○	○		**&**					○	0	○	○		*	& (and)
’			○			0	○	○	○							0				*	Apostrophe
*****	○		○		○	0		○								0				*	Asterisk
,	○		○		○	0	○			**,**			○	○	○	0		○	○	*	Comma
;	○		○	○	○	0		○	○							0				*	Semicolon
<			○	○	○	0	○									0				*	Left angular bracket
=	○		○	○	○	0	○		○							0				*	Equal sign
>	○		○	○	○	0	○	○								0				*	Right angular bracket
?			○	○	○	0	○	○	○							0				*	Question mark
@	○	○				0										0				*	@ sign
”			○			0		○								0				*	Quotation mark
[	○	○		○	○	0		○	○							0				*	Left brace
]	○	○		○	○	0	○		○							0				*	Right brace

(Note 1) “*” code is read in memory only it appears in a note and is invalid in other information areas.

(Note 2) “?” code is read in memory only it appears in a note and causes alarm when it appears in other information areas.

(Note 3) When a user macro is selected, the following codes are usable in the effective information areas.

For ISO: +, [,], #, *, =, B, C, H, J, V, Y.

For EIA: +, [,], &, the codes set by parameter and B, C, H, J, V, Y.

(Note 4) The codes not listed in the table are ineffective even its parity is correct.

(Note 5) The use of a code with incorrect parity may cause TH alarm. But it is ignored in notes and does not cause TH alarm.

(Note 6) The output of all the eight digits does not cause alarm for EIA code.

Appendix 2 G Codes List

G Codes	Group No.	Function
G00*	**01**	Positioning (quick)
G01*		Linear interpolation (cutting feed)
G02		Arc interpolation CW (clockwise circle)
G03		Arc interpolation CCW(counterclockwise circle)
G04	**00**	Suspension
G07		imaginary axis designation
G09		Accurate stop check
G10		Offset, work zero offset setting
G17*	**02**	To select XY plane
G18		To select ZX plane
G19		To select YZ plane
G20	**06**	To input in inch system
G21		To input in metric system
G22	**04**	Storage travel limit ON
G23		Storage travel limit OFF
G27	**00**	Reference point return check
G28		To return to reference point
G29		To return from reference point
G30		To return to the 2^{nd}, 3^{rd} and 4^{th} reference points
G31		To skip over cutting
G33	**01**	Thread cutting
G40*		To cancel tool compensation
G41		Tool compensation – left side
G42		Tool compensation – right side
G43*	**08**	Tool radius compensation, +
G44*		Tool radius compensation, -
G49*		To cancel tool length compensation
G45	**00**	To increase tool offset
G46		To reduce tool offset
G47		To increase tool offset by twice
G48		To reduce tool offset by twice
G50*	**11**	Zooming OFF
G51		Zooming ON
G54*	**14**	To select workpiece coordinate system 1
G55		To select workpiece coordinate system 2
G56		To select workpiece coordinate system 3
G57		To select workpiece coordinate system 4
G58		To select workpiece coordinate system 5
G59		To select workpiece coordinate system 6
G60	**00**	One-way positioning
G61	**15**	Accurate stop detecting mode
G62		To enable automatic angle adjustment
G64*		Continuous cutting mode
G65	**00**	Simple recall of a user macro
G66	**2**	Modal recall of microinstruction
G67*		To cancel modal recall of microinstruction
G73	**09**	Gun drilling cycle
G74		Left-handed tapping cycle
G76		Finish boring

G80*		To disable fixed cycle
G81		Drilling cycle, spotter
G82		Drilling cycle, counter boring
G83		Depth drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Reverse boring cycle
G88		Boring cycle
G89		Boring cycle
G90*	**03**	Absolute-value programming
G91*		Incremental-value programming
G92	**00**	Coordinate system setting
G94*	**05**	Feed per minute
G95		Feed per rotation
G96	**13**	Feed at constant speed
G97*		To cancel constant speed control
G98*	**10**	To return fixed cycle to the initial point
G99		To return fixed cycle to point R

(Note 1) The G codes marked with * are the initial G codes of all groups. That is, these G codes are established when power on or pressing the RESET key (the system parameters specifying initial G codes are validate).

The selection of the state of the initial G codes such as G00, G01, G43, G44, G49, G90, G91 or G94 and G95 shall be set by parameters (G00, G43, G44, G90 and G95).

For G20 or G21, it becomes the state before power off.

(Note 2) The G codes in group 00 are modal and are only valid in the blocks they belong to.

(Note 3) When a G code not listed in the above table is specified or an optional G code not defined by control unit is specified, (N0.010) will give an alarm. (N0.010) But G38 and G39 are ignored.

(Note 4) Some G codes may be specified in the same block even they do not belong to the same group. When 2 or more G codes than belong to the same group are specified in a block, the lastly specified G code will be valid.

(Note 5) If any G code in group 01 in fixed cycle mode, the fixed cycle will be automatically disabled and the system will be in G80 state. However, the G codes in group 01 are not subject to the influence of the G codes of fixed cycle.

(Note 6) G70 and G71 replaces G20 and G21 (special G codes) by parameter setting (GSP).

(Note 7) The G codes of all groups are displayed.

Appendix 3 Ranges of Instruction Values

		Input in mm Output in mm	Input in inch Output in mm	Input in mm Output in inch	Input in inch Output in inch
Minimum setting unit		0.001mm 0.001°	0.0001inch 0.001°	0.001mm 0.001°	0.0001inch 0.001°
Maximum travel (the value starts from reference point)		±99999.999mm	±99999.999mm	±3937.0078inch	±99999.999inch
Max. instruction value		±99999.999mm ±99999.999°	±3937.0078inch ±99999.999°	±99999.999mm ±99999.999°	±99999.999inch ±99999.999°
When adjusting cutting feed by 100%	Feed per minute	1 mm/min to 15000 mm/min	0.01 inch/min to 600.00 inch/min	1 mm/min to 15000 mm/min	0.01 inch/min to 600.00 inch/min
Feedrate at high speed (all axes are independent)		30 mm/min to 15000 mm/min	30 mm/min to 15000 mm/min	3.0 inch/min to 600.0 inch/min	3.0 inch/min to 600.0 inch/min
Upper limit of cutting feedrate		6 mm/min to 15000 mm/min	6 mm/min to 15000 mm/min	0.6 inch/min to 600.0 inch/min	0.6 inch/min to 600.0 inch/min
Manual feedrate					
F0					
JOG feedrate		1 mm/min to 2000 mm/min	0.04 inch/min to 78.7 inch/min	0.5 mm/min to 1016 mm/min	0.02 inch/min to 40 inch/min
Coordinates of 2nd reference point (the value starts from reference point)		0 mm to ± 99999.999mm	0 mm to ± 99999.999mm	0 inch to ±3937.0078inch	0 inch to ±99999.999inch
Tool offset		0 mm to ± 99999.999mm	0 inch to ± 99999.999inch	0 mm to ±99999.999mm	0 inch to ±99999.999inch
Minimum value of incremental feed		0.001mm	0.0001inch	0.001mm	0.0001inch
Clearance compensation		0mm to 0.255mm	0 mm to 0.255mm	0 inch to 0.255inch	0 inch to 0.255inch
Compensation of screw pitch error		0mm to ±0.007mm	0mm to ±0.007mm	0inch to ±0.007inch	0inch±0.007inch
Setting range of storage travel limit (the value starts from reference point)		0mm to ± 99999.999mm	0mm to ± 99999.999mm	0inch to ±3937.0078inch	0inch to ±99999.999inch
Pause		0s to 99999.999s	0s to 99999.999s	0s to 99999.999s	0s to 99999.999s

Appendix 4 Calculating Chart

A4.1 The tool path in a turning angle section

(1) Summary

Due to the delay of servo system (as a result of exponential deceleration/acceleration during cutting or the use of the positioning system of a DC servo motor), a minor deviation exists between the tool path (tool center path) and instructed path as shown in Fig. 4.1 (a). In this center, the time constant T1 of exponential deceleration/acceleration is fixed to 0.

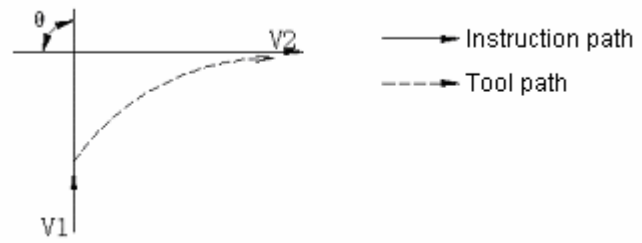


Fig. 4.1 (a): The tool path regarding a turning angle section

The tool path depends on the following parameters:

(a) Feedrate (V_1, V_2)

(b) Degree of the turning angle (θ)

(c) Time constant (T_1)(T_1 = 0) of exponential deceleration/acceleration during cutting

(d) Loop gain of positioning system

(e) Existence of buffer

This document makes theoretical analysis of tool path with the above parameters. The tool path of temporarily set parameters is displayed in graph.

The above notes shall be observed for programming. Note whether the machining shape is within the desired accuracy range.

That is, when it is possible to reach the theoretical accuracy, hold for a proper time using the suspension function until the instruction speed drops to zero before reading in the next block.

(2) Analysis

In the following conditions, make analysis of the tool path as shown in Fig. 4.1(b).

(a) Feedrate remains constant in the blocks before and after the turning angle.

(b) The control is provided with a buffer (error changes with the reading rate of the input equipment and the number of characters of the next block).

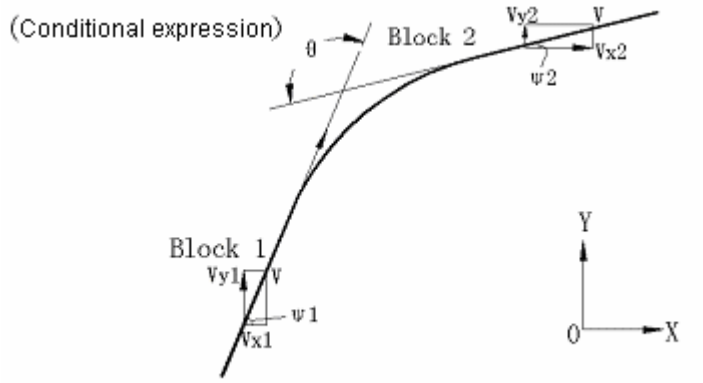


Fig. 4.1 (b): Instruction

$V_X1 = V \cdot \cos\varphi1$

$V_Y1 = V \cdot \sin\varphi1$

$V_X2 = V \cdot \cos\varphi2$

$V_Y2 = V \cdot \sin\varphi2$

$\pi - (\varphi1 - \varphi2) = \theta$

[Explanations for the symbols]

V: Feedrate in the blocks before and after turning angle

V_X1: X component of the feedrate in the previous block

V_Y1: Y component of the feedrate in the previous block

V_X2: X component of the feedrate in the next block

V_Y2: Y component of the feedrate in the next block

θ: Angle of the turning angle

φ1: The included angle between the direction of the instruction path in the previous block and axis X

φ2: The included angle between the direction of the instruction path in the next block and axis X

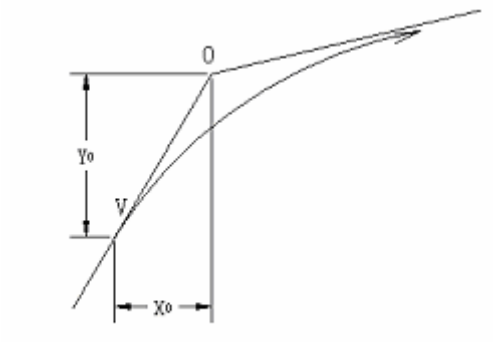


Fig. 4.1 (C): initial value

Calculation of initial value

The initial value at the beginning of a turning angle, i.e. the X and Y coordinates at the end of the distribution of the control unit are determined by the time constant of the positioning system of the used DC motor.

$X_0 = Vx_1(T_1+T_2)$……(1) $Y_0 = Vy_1(T_1+T_2)$……(2)

T_1: Time constant ($T_1 = 0$) of exponential deceleration/acceleration

T_2: Time constant (reciprocal of position loop gain) of positioning system

Analysis of tool path

The feedrates in the directions of axis X and Y in the turning angle are expressed in the following formulas.

$$Vx(t)=(Vx_2-Vx_1)[1-\frac{Vx1}{T_1-T_2}\{T_1\times\exp\left(-\frac{t}{T_1}\right)-T_2\times\exp\left(-\frac{t}{T_2}\right)\}+Vx1]$$

$$=Vx_2\left[1-\frac{Vx1}{T_1+T_2}\left\{T_1\times\exp\left(-\frac{t}{T_1}\right)-T_2\times\exp\left(-\frac{t}{T_2}\right)\right\}\right] \quad \text{……(3)}$$

$$Vy(t)=\frac{Vy_1-Vy_2}{T_1-T_2}\left\{T_1\times\exp\left(-\frac{t}{T_1}\right)-T_2\times\exp\left(-\frac{t}{T_2}\right)\right\}+Vy_2 \quad \text{……(4)}$$

The coordinates of the tool path at time t are calculated with the following formula.

$$X(t)=\int_0^t Vx(t)dt-X_0$$

$$=\frac{Vx_2-Vx_1}{T_1-T_2}\left\{T_1^2\times\exp\left(-\frac{t}{T_1}\right)-T_2^2\times\exp\left(-\frac{t}{T_2}\right)\right\}-Vx_2(T_1+T_2-t) \quad \text{……(5)}$$

$$Y(t)=\int_0^t Vy(t)dt-Y_0$$

$$=\frac{Vy_2-Vy_1}{T_1-T_2}\left\{T_1^2\times\exp\left(-\frac{t}{T_1}\right)-T_2^2\times\exp\left(-\frac{t}{T_2}\right)\right\}-Vy_2(T_1+T_2-t) \quad \text{……(6)}$$

A4.2 Error of radius direction during arc cutting

When using a DC servo motor, a delay is produced between the input and output axes due to the factors of the positioning system. The linear interpolation on instruction path does not cause any error. Arc interpolation, especially the arc cutting at high speed may cause a deviation in the direction of radius, which is determined with the following formula.

△Y: Maximum radius error (mm)

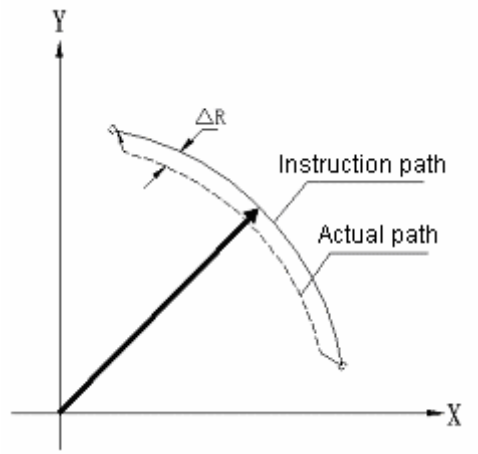
V: Feedrate (mm/sec)

r: Arc radius (mm)

T_1: Time constant (sec) (T_1 = 0) of exponential deceleration/acceleration during cutting

T_2: Time constant (sec) (reciprocal of position loop gain) of positioning system

$$\triangle \quad r = \frac{1}{2}\left(T_1^2 + T_2^2\right) \times \frac{V_2}{r} \quad \text{..... (1)}$$



The radius r(mm) and permissible error △r(mm) of a workpiece are given in actual machining. Permissible speed limit V(mm/sec) can be calculated with the formula (1).

The Time constant of deceleration/acceleration during cutting set by the unit varies with machine types. Refer to the user manual supplied with the machine.

Appendix 5 Parameters

Display and setting procedures of parameters

When the NC is connected to a servo motor or the machine, the specifications and functions of the machine shall be set to maximize the performance of the servo motor. Since its contents vary with machine types, please refer to the additional parameters of the machine. End user is not allowed to change the parameters.

A5.1 Display of parameter

(a) Press the PARAMETER key.

(b) Select the desired page with the PAGE key, or N , parameter number and INPUT

A5.2 Setting of parameter

Setting in MDI mode

(a) Switch the PRM.WT switch on the main unit to ENABLE (ON) to activate the warning indicator on the panel.

(b) Set the MDI mode.

(c) Press the PARAMETER key.

(d) Press the N , the parameter number to be set and INPUT. The cursor will appear underneath the number provided that the page of the parameter number to be set is selected. (This may also be done with the PAGE button and cursor ↓ or ↑ button.)

(e) Set with P , set data and INPUT.

In the event of error during typing in, press the CANCEL key.

(f) Make sure the setting is correct.

(g) After the setting of all parameters and end of confirmation, return the switch on the main PCB to DISABLE (OFF).

(h) To terminate the alarm state (No. 100), press the RESET button.

A5.3 Parameters list

Parameter No.

	0	0	6

ORWD	SCTO	EENB	OTCS	FMIC	MDL	MIC	SCW
7	6	5	4	3	2	1	0

ORWD: Retention parameter

SCTO 1: Perform detection of speed reaching signal during instruction and changing from quick speed to cutting feed.

0: Not perform detection of speed reaching signal.

EENB 1: Servo OFF signal is active.

0: Servo OFF signal is inactive.

OTCE 1: Stop once the travel switch of hardware is pressed.

(Mechanical position is lost)

0: Slow down and stop once the travel switch of hardware is pressed. (Mechanical position is not lost)

FMIC 1: When inputting in metric system, the unit of feedrate is 1/10.

0: Not 1/10

MDL 1: The minimum unit of the indication on the position display unit is 0.01mm or input in metric system and 0.001inch for input in inch system.

0: The minimum setting unit of the indication on the position display unit is 0.001mm or input in metric system and 0.001inch for input in inch system.

MIC 1: The minimum setting unit of the indication on the position display unit is 0.01mm or input in metric system and 0.0001inch for input in inch system.

0: The minimum setting unit of the indication on the position display unit is 0.001mm or input in metric system and 0.0001inch for input in inch system.

SCW 1: The minimum travel unit is 0.0001inch (machine in inch system)

0: The minimum travel unit is 0.001mm (machine in metric system)

	0	0	7	ADFT	EOM	CINP	DCS	CLER	TVC	PPD	RDRN
				7	6	5	4	3	2	1	0

ADET 1: Perform automatic drift compensation

0: Not perform automatic drift compensation.

EOM 1: If M30 is sent to the machine side and FIN returns when M30 is instructed, it is continuously executed from the preceding block of the program, or the machine side does not send back FIN signal but external reset signal. The program returns to the beginning to enter into reset state (in automatic mode).

0: When M30 is instructed, only M30 is sent out to the machine side and the program does not return to the beginning unless reset and rewinding signal is used (in automatic mode).

CINP 1: Instruction speed reduces to 0 between two non-cutting blocks. The next block is not continued until the machine confirms that the designated position is reached (the confirmation is called positioning detection).

0 The next block is executed once the instruction speed reduces to 0 between two non-cutting blocks (not perform positioning detection).

DCS 1: The start button on the MDI panel does not pass through the machine side, but directly is switched on the NC side (only for MDI mode).

0: The start button on the MDI panel is sent to the machine side, and then the start button on the machine side returns for start.

CLER 1: The NC is brought into clearing state (see Appendix 7 for clearing state) using the RESET button, external reset signal, reset and rewinding signal.

0: The NC is brought into reset state using the RESET button, external reset signal, and reset and rewinding signal.

TVC 1: Not perform TV check in the control output section (note section)

0: Perform TV check in the control output section (note section)

PPD 1: Preset the position display unit by setting a coordinate system.

0: Not preset the position display unit during the setting of a coordinate system.

RDRN 1: Dry run is also valid for rapid feed instruction.

0: Dry run is not valid for rapid feed instruction.

	0	0	8	ICR		GSP	G44	G90	G95	G43	G00
				7	6	5	4	3	2	1	0

ICR 1: When ISO code is used for output, EOB will be output by LF.

0: When ISO code is used for output, EOB will be output by LFCR CR.

GSP 1: Use a special G code.

0: Use a standard G code.

G90 1: Enter into G90 mode when in power on and clearing state.

0: Enter into G91 mode when in power on and clearing state.

G95 1: Enter into G95 mode when in power on and clearing state.

0: Enter into G94 mode when in power on and clearing state.

G00 1: Enter into G00 mode when in power on and clearing state.

0: Enter into G01 mode when in power on and clearing state.

G44, G43

G44	G43	Initial state
1	0	Enter into G44 mode when in power on and clearing state.
0	1	Enter into G43 mode when in power on and clearing state.
0	0	Enter into G49 mode when in power on and clearing state.

	0	0	9	FIX2	RWL	MCF	FMFS	FCUT	ILVL	EFR1	TDRN
				7	6	5	4	3	2	1	0

FIX2 1: Output the M code in a fixed cycle (fixed cycle II).

0: Not output the M code but SSP and SRV in a fixed cycle (fixed cycle II)

RWL 1: Store the outside of the travel limit 2 as a prohibited area.

0: Store the inside of the travel limit 2 as a prohibited area.

MCF 1: Output EF at the end of the positioning of G81 (axis Z does not move).

0: Output EF at the end of the positioning of G81 (axis Z moves).

FMFS 1: Output FMF signal twice in a fixed cycle.

0: Output FMF signal once in a fixed cycle.

FCUT 1: The travel of the axes X and Y in a fixed cycle uses the G codes in Group 01.

0: The travel of the axes X and Y in a fixed cycle usually uses rapid feed.

ILVL 1: Change the initial plane points with the RESET button.

0: Not change the initial plane points even using the RESET button.

EFRI 1: Output EF with a photoelectric coupler

0: Output EF with a relay.

TDRN 1: Also valid for dry run of thread cutting.

0: Invalid for dry run of thread cutting.

	0	1	0	TCW	CWM	SOV	SLCC	OFSD	SOVC		ISOT
				7	6	5	4	3	2	1	0

Signs for TCW, CWM and S4-digit output

TCW	CWM	Sign
0	0	M03 M04 are all positive.
0	1	M03 M04 are all negative.
1	0	M03 is positive while M04 is negative.
1	1	M03 is negative while M04 is positive.

SOV 1: Enable main axis adjustment.

0: Disable main axis adjustment.

TLCC 1: The change of offset in G43 and G44 modes starts to take effect from the next block.

0: The change of offset in G43 and G44 modes starts to take effect from the following H and D codes.

OFSD 1: Tool position offset (G45 to G48) uses D code.

0: Tool position offset (G45 to G48) uses H code.

SOVS 1: Spindle adjustment is clamped at 100% in tapping.

0: Spindle adjustment is not clamped at 100% even in tapping.

ISOT 1: With the storage type travel limit selection, JOG rapid feed is valid even returning to the reference point is not performed.

0: With the storage type travel limit selection, JOG rapid feed is valid unless returning to the reference point is performed.

	0	1	1	DGNE	SETE	DECI	SSPB		VCT	SUPM	ADLN
				7	6	5	4	3	2	1	0

DGNE 1: Enable data output in diagnosis.

0: Disable data output in diagnosis.

SETE 1: Input setting is possible when the lock key is closed.

0: Input setting is not possible when the lock key is closed.

DECI 1: Decelerate when the decelerating signal is “1” during the return to reference point.

0: Decelerate when the decelerating signal is “0” during the return to reference point.

SSPB 1: Spindle stops when the spindle stop input signal (SSP) is “o”.

0: Spindle stops when the spindle stop input signal (SSP) is “1”.

VCT 1: It is possible to specify tool compensation vector with I, J and K.

0: It is impossible to specify (general automatically calculate).

SUPM 1: Start and cancel type B in tool radius compensation C.

0: Start and cancel type A in tool radius compensation C. (See this document for type A/B.)

ADLN 1: The 4th axis is used as a linear axis.

0: The 4th axis is used as an axis of revolution.

	0	1	2	ZGM4	ZGGMZ	ZGMY	ZGMX	ZM4	ZMZ	ZMY	ZMX
				7	6	5	4	3	2	1	0

ZGMX, ZGMY, ZGMZ and ZGM4 are the return-to-reference-point modes for axes X, Y and Z and the 4th axis respectively.

1: magnetic switch mode 0: grid mode

ZMX, ZMY, ZMZ and ZM4 are the return-to-reference-point directions as well as the initial directions of clearance when power on for axes X, Y and Z and the 4th axis respectively.

1: The return-to-reference-point direction as well as the initial direction of clearance is negative.

0: The return-to-reference-point direction as well as the initial direction of clearance is positive.

(Note 1) For an axis with return-to-reference-point function, the return-to-reference-point direction is identical with the initial direction of clearance. For an axis without return-to-reference-point function, the parameter only has the meaning of initial direction of clearance.

(Note 2) After power on, clearance compensation is made when moving in the direction set by this parameter.

	0	1	3

PSG2	PSG1		PHS				
7	6	5	4	3	2	1	0

PSG2, PSG The tooth to spindle ratio of spindle and position coder

Override	PSG2	PSG1
X1	0	0
X2	0	1
X4	1	0
X5	1	1

$$\text{Override} = \frac{\text{Number of spindle revolutions}}{\text{number of position coder revolutions}}$$

When in PHS and PSCG modes (rotary transformer and inductosyn), the initial setting of relative offset:

1: does not automatically set phase deviation

0: automatically sets phase deviation. PHS automatically changes to “1” after the setting is performed once.

0 1 4

		DMR	X		GRD	X	
7	6	5	4	3	2	1	0

0 1 5

		DMR	Y		GRD	Y	
7	6	5	4	3	2	1	0

0 1 6

		DMR	Z		GRD	Z	
7	6	5	4	3	2	1	0

0 1 7		DMR 4		GRD 4	
	7	6	5	4	3	2	1	0

4 1 0		DMR 5		GRD 5	
	7	6	5	4	3	2	1	0

DMRX, DMRY, DMRZ, DMR4 and DMR5 are the measuring override ratio for axes X, Y, Z and the 4th axis respectively.

Setting codes			Overrides	
			Pulse coder	Rotary transformer and inductosyn
0	0	0	1/2	1/8
0	0	1	1	1/4
0	1	0	1	1/4
0	1	1	2	1/2
1	0	0	3/2	3/8
1	0	1	3	3/4
1	1	0	2	1/2
1	1	1	4	1

GRDX, GRDY, GRDZ, GRD4 and GRD5 are the capacities of the reference counters for axes X, Y, Z and the 4th axis respectively.

Setting codes				Capacity of a cycle
0	0	0	1	2000
0	0	1	0	3000
0	0	1	1	4000
0	1	0	0	5000
0	1	0	1	6000
0	1	1	1	8000
1	0	0	1	10000

(Note) The codes other than those listed in the above table are treated as 8000 capacity.

0 1 8			CPF2	CPF1
	3	2	1	0

Pulse frequency (holoaxial calling) for CPF2 and CPF1 clearance compensations.

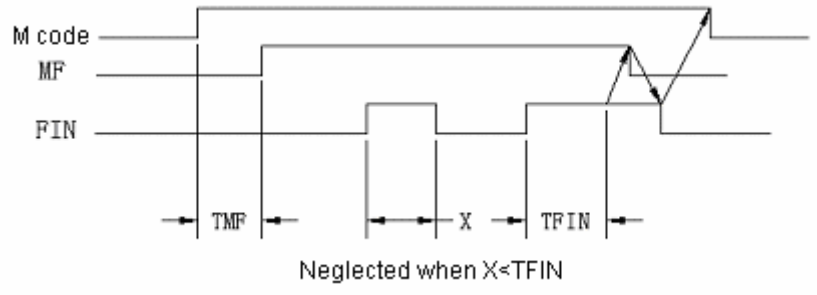
Frequency, KHZ	CPF2	CPF1
32	0	0
64	0	1
128	1	0
256	1	1

(Must be set to 256KHZ)

0 1 9	TMF	TFIN
	7 6 5 4	3 2 1 0

TMF The time from the send-out of M, S, T and B codes to that of MF, SF, SF and TFBF: 16—126ms (16ms interval).

TFIN The time of FIN signal receiving width: 16—265ms (16ms interval).



T M F	T F I N	Setting			
15m sec	>15m sec	0	0	0	0
32m sec	>15m sec	0	0	0	1
48m sec	>15m sec	0	0	1	0
64m sec	>15m sec	0	0	1	1
80m sec	>15m sec	0	1	0	0
96m sec	>15m sec	0	1	0	1
112m sec	>15m sec	0	1	1	0
128m sec	>15m sec	0	1	1	1
144m sec	>15m sec	1	0	0	0
160m sec	>15m sec	1	0	0	1
176m sec	>15m sec	1	0	1	0
192m sec	>15m sec	1	0	1	1
208m sec	>15m sec	1	1	0	0
224m sec	>15m sec	1	1	0	1
240m sec	>15m sec	1	1	1	0
256m sec	>15m sec	1	1	1	1

	0	2	0

CLSI			ZTN5	ZTN4	ZTNZ	ZTNY	ZTNX
7	6	5	4	3	2	1	0

CLSI 1: Not detect servo position circuit LSI.

0: Detect servo position circuit LSI.

ZTNX, Y, Z, 4 and 5 are the availability of return-to-reference-point function for axes X, Y and Z as well as the 4th and 5th axis respectively.

1: With return-to-reference-point function

0: Without return-to-reference-point function

	0	2	1

G84S	SFOU	EDMZ	EDMY	EDMX	EDPZ	EDPY	EDPX
7	6	5	4	3	2	1	0

G84S 1: When S 12-digit output A and S analog output A are used in fixed cycles G74 and G84, method B is valid.

0: When S 12-digit output A and S analog output A are used in fixed cycles G74 and G84, method A is valid (see connection guides for method A/B) .

SFOU For 12-digit outputs A and B as well as analog output A, SF setting is not output in the output without gear switching.

1: Output SF.

0: Not output SF.

EDMX, EDMY and EDMZ correspond to the negative instructions of axes X, Y and Z respectively.

1: Valid for rapid feed and external deceleration of cutting feed

0: Only valid for rapid feed.

EDPC, EDPY and EDPZ correspond to the positive instructions of axes X, Y and Z respectively.

1: Valid for rapid feed and external deceleration of cutting feed

0: Only valid for rapid feed.

	0	2	2

	SIJ	PMXY2	PMXY1	RS43	FXCD	TAPSG	FXCS
7	6	5	4	3	2	1	0

SIJ Setting of tool returning method in fixed cycle G76 or G87

1: Set direction and amount of travel with address I and J.

0: The amount of travel uses address Q as instruction and its direction is determined by parameters PMXY1 and PMXY2

PMXY2, 1 Setting of tool returning direction in fixed cycle G76 or G87(only valid when SIJ =0)

PMXY2	PMXY1	Returning direction
0	0	+X
0	1	−X
1	0	+Y
1	1	−Y

RS43 1: The offset vectors of G43 and G44 are retained after reset.

0: The offset vectors of G43 and G44 are cleared after reset.

FXCD 1: Dwell instruction is valid in fixed cycles G74 and G84.

0: Dwell instruction is invalid in fixed cycles G74 and G84.

TAPSG 1: Output tapping signal in fixed cycles G74 and G84.

(Valid only when FLX2 = 1).

0: Not output tapping signal.

FXCS 1: M05 is not output and spindle rotates reversely or forward in fixed cycles G74 and G84 (Valid only when FIX2 = 1)

0: M05 is output and spindle rotates reversely or forward in fixed cycles G74 and G84

	0	2	3

	EXIOD						
7	6	5	4	3	2	1	0

EXIOD 1: To multiply the unit of external origin offset by 10.

0: Not to multiply it by10.

	0	2	4	PML2	PML1	DLME	RDAL	ADW2	ADW1	ADW0	FMT
				7	6	5	4	3	2	1	0

PMC2, 1 Screw pitch error compensation override

The set compensation is output after it is multiplied by the override.

FMT 1: Perform software parity check for part program edit area.

0: Not perform the check.

When it is ready for the check, the parameter is automatically set to 1. It cannot be externally switched on or off.

PML2	PML1	Override
0	0	×1
0	1	×2
1	0	×4
1	1	×8

DLME 1: All programs stored before are automatically cleared when storing programs in memory.

0: Not automatically clear.

RDAL 1: Generally all programs are stored when storing programs in memory.

0: When storing programs in memory, storing a program or all programs is determined by MDI operations.

ADW2, 1, 0 Select the name for the 4th axis in the even of data output or display on LCD.

ADW2	ADW1	ADW0	Address
0	0	0	A
0	0	1	B
0	1	0	C
0	1	1	U
1	0	0	V
1	0	1	W

	0	2	5	MUSR	MCYL	MSUB	MPRM				TSE
				7	6	5	4	3	2	1	0

MSUB 1: Subprogram type user macro terminates.

0: Macro type user

MURS 1: Use user macro look-at-me function

0: Not use.

MCYL 1: User macro terminates in cycle operations.

0: User macro does not terminate in cycle operations

(Note) The local variable for a macro terminated and called by subprogram type user macro does not change. The local variable for a macro terminated and called by macro type user macro differs from that is used in the current program.

MPRM 1: That a user macro terminates and controls M codes is set by parameter.

0: The look-at-me of a user macro is controlled by M96 and M97.

TSE 1: The look-at-me of a user macro adopts state triggering mode.

0: Adopt fringe triggering mode.

	0	2	6			NGMP	OFFVY		OGE		CKIM

7 6 5 4 3 2 1 0

MGMP The amounts of travel of manual pulse generator are as follows.

NGMP	MP2	MP1		Amounts of travel
0	0	0		0.001mm/0.0001inch
0	0	1		0.01mm/0.001inch
0	1	0		0.1mm/0.01inch
0	1	1		0.1mm/0.01inch
1	0	0		0.01mm/0.001inch
1	0	1		0.001mm/0.0001inch
1	1	0		0.1mm/0.01inch
1	1	1		0.1mm/0.01inch

OFFVY 1: Servo alarm is not given even when VRDY is ON before PRDY output.

0: Servo alarm is not given when VRDY is ON before PRDY output.

CKIM 1: Ignore the switching of signal during automatic operation (the state of during CYCLE START is valid).

0: Immediately enable machine lock signal.

(Note) Machine lock is always active in JOG mode.

1: Storage sorting is not performed during program search.

0: Storage sorting is performed during program search.

	0	2	7	CMRX

	0	2	8	CMRY

	0	2	9	CMRZ

	0	3	0	CMR4

	4	1	4	CMR5

CMRX, CMRY, CMRZ, CMR4 and CMR5 are the instruction multiplying powers for axes X, Y and Z as well as the 4th and 5th axes respectively.

1	0.5
2	1
4	2
10	5
20	10

	0	3	1	VLOCX

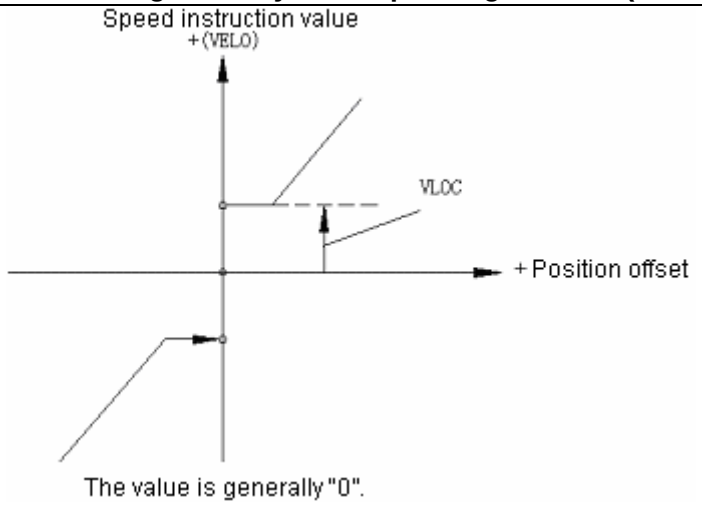
	0	3	2	VLOCY

	0	3	3	VLOCZ

	0	3	4	VLOC4

	4	1	5	VLOC5

VLOCX, Y, Z, 4 and 5 are the minimum values of speed instruction for axes X, Y and Z as well as the 4th and 5th axes respectively.



	0	3	5	MBUF1

	0	3	6	MBUF2

MBUF1, MBUF2 Up to 2 settings between 00 and 97 may be set for the M codes without buffer.

	0	3	7	SPGST

SPGST The setting between 0 and 255 for the number of revolutions of spindle motor (S12 or S analog output A/B) during the shifting of spindle gear.

$$\text{Setting=}\frac{\text{Number of revolutions of spindle motor during the shifting of gear}}{\text{Maximum number of revolutions of spindle motor}}\times 4095$$

	0	3	8	SPSOR

SPSOR Number of revolutions during accurate stop of spindle (S12 or S analog output A/B)

Settings 0 to 255 Unit: rpm

	0	3	9	PECZRX

	0	4	0	PECZRY

	0	4	1	PECZRZ

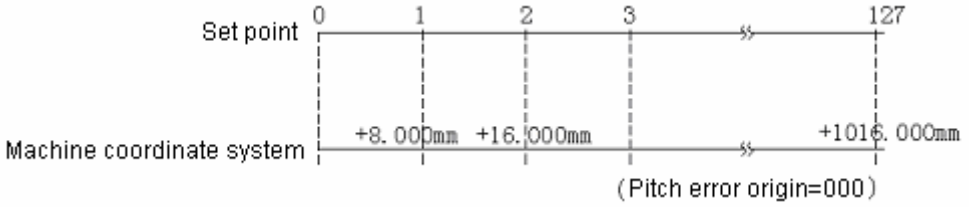
	0	4	2	PECZR4

	4	1	6	PECZR5

PECZRX to 5 are the settings of pitch error origin for axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to 127

Set some points corresponding to the reference point. For example, assuming that the pitch error origin is set to 0 and the set point 1 is located at +8.000mm, then the compensation range of the set point 127 at 1016.000mm is 0 mm to 1016.000mm.



This is the occasion that pitch error compensating clearance (parameters 163 to 166) is 8000.

0	4	3	UMMCD4
0	4	4	UMMCD5
0	4	5	UMMCD6
0	4	6	UMMCD7
0	4	7	UMMCD8
0	4	8	UMMCD9
0	4	9	UMMCD10
0	5	0	UMMCD11
0	5	1	UMMCD12
0	5	2	UMMCD13

UMMCD4 to UMMCD13
Up to 10 M codes for calling user macros can be set (assignment of independent variable is also possible).

Settings 01 to 97

0	5	3	MACINTON

MACINTON: The code for a user macro to terminate its active state.

Settings 03 to 97

The parameters are only valid when the parameter 025—MPRM = 1.

0	5	4	MACINTOF

MACINTON: The code for a user macro to terminate its inactive state.

Settings 03 to 97

The parameters are only valid when the parameter 025—MPRM = 1.

0	5	7	TMHOR (hr)
0	5	8	TMMIN (min)
0	5	9	TMSEC (sec)

TMHOR Display in hour: 0 to 255 (increment: 1 hr)

TMMIN Display in minute: 0 to 59 (increment: 1min) Display during machining time

TMSEC Display in sec: 0 to 58 (increment: 2 sec)

Machining time (when the STL indicator is lit) is displayed in hour, minute or sec by setting.

Machining time is saved in nonvolatile memory even in the event of power failure.

To preset it to zero, proceed as the setting operations for switching on (also for settings).

	0	6	0	IDXUNT

IDXUNT Minimum indexing angle of rotary workbench

Settings 1 to 255 Unit: deg(degree)

It is only valid when it is used in junction with the parameter 3/4—5(IFIX)= 1. Now P/S alarm (No. 180) will be given if the travel other than the integral multiples of IDXUNT setting is specified.

In addition, P/S alarm (No. 180) will be given if the instructed value of coordinate setting (G92), the parameter value of automatic coordinate system setting and the parameter value of workpiece origin offset are beyond the integral multiples set in IDXUNT.

	0	6	1	FIDN

FIND The constant of change in feedrate when the pulse generator is moved for one division by JOG under F1-digit instruction.

△ $F = \frac{F\max_i}{100n}$ (i=1.2)

Set the n in the above formula, namely the feedrate hits $Fmax_i$ when the pulse generator is rotated for n turns by JOG.

Settings 1 to 12

The $Fmax_i$ in the above formula is the upper limit of the feedrate of F1-digit instruction and is set by parameter No.065, 066.

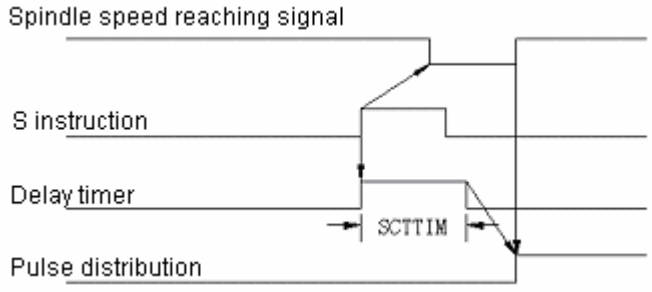
Fmax1 Upper limit of the feedrate of F1 to F4 (065)

Fmax2 Upper limit of the feedrate of F5 to F9 (066)

	0	6	2	SCTTIM

SCTTIM The delay time setting of the spindle speed reaching signal detection; the setting of the time from the start of execution of S function to the end of spindle speed reaching signal detection.

Settings 0 to 255 Unit: msec



	0	6	3				ZDL5	EX5NG	HIRS	AP5	G605
				7	6	5	4	3	2	1	0

ADL5 1: The 5th axis is a linear axis.

0: The 5th axis is an axis of rotation

HIR5 1: Enable the look-at-me of the 5th axis handwheel.

0: Disable the look-at-me of the 5th axis handwheel.

EX5NG 1: Enable the 5th axis neglecting signal.

0: Disable the 5th axis neglecting signal.

AP5 1: Enable the automatic coordinate system setting for the 5th axis.

0: Disable.

G605 1: The approaching direction of the single-directional positioning for the 5th axis is negative.

0: The approaching direction is positive.

	0	6	4						SCLZ	SCLY	SCLX
				7	6	5	4	3	2	1	0

SCLX, SCLY and SCLZ are the settings of availability of the zooming functions of axes X, Y and Z respectively.

1: Enable the zooming function.

0: Disable the zooming function.

	0	6	5	FIDMAX1

	0	6	6	FIDMAX2

FIDMAX1 , FIDMAX2 The upper limit of the feedrate of F1-digit instruction

FIDMAX1 The upper limit of the feedrate of F1 to F4

FIDMAX2 The upper limit of the feedrate of F5 to F9

Settings 0 to 15000 Unit: mm/min (output in metric system)

0 to 6000 Unit: 0.1 inch/min(output in inch system)

See parameter No.061 for details.

	0	6	7	CYCR

CYCR The setting of return in fixed cycle G73 (high-speed depth drilling cycle)

Settings : Input 0 to 32767 in metric system, unit: 0.001mm.

Input 0 to 32767 in inch system, unit 0.0001inch.

It is also to set with specified values.

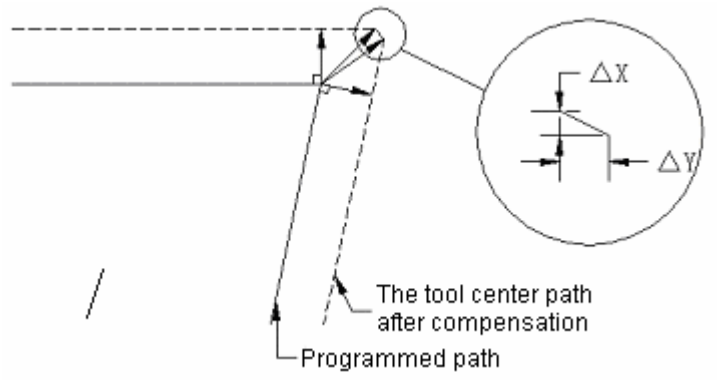
	0	6	8	CYCD

	0	6	9	CRCDL

CRCDL The neglecting limit of the minor amount of travel while the tool is machining on the exterior side of the turning angle near 90° during tool radius compensation.

Settings: Input 0 to 16383 in metric system, unit: 0.001mm

Input 0 to 16383 in inch system, unit: 0.0001inch



When △X＜CRCDL and △Y＜CRCDL, minor travel is neglected. In this way the influence as a result of workpiece's stop on workpiece may be prevented.

	0	7	0	INPX
	0	7	1	INPY
	0	7	2	INPZ
	0	7	3	INP4
	4	2	5	INP5

INPX, INPY, INPZ, INP4 and INP5 are the positioning width settings for axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to 32767 measurement units

	0	7	4	STPEX
	0	7	5	STPEY
	0	7	6	STPEZ
	0	7	7	STPE4
	4	2	6	STPE5

STPEX, STPEY, STPEZ, STPE4 and STPE5 are position deviation limits for axes X, Y and Z as well as the 4th and 5th axes respectively.

Setting units: 0 to 32767 measurement units

	0	7	8	SERRX
	0	7	9	SERRY
	0	8	0	SERRZ
	0	8	1	SERR4
	4	2	7	SERR5

SERRX, Y, Z, 4 and 5 are position deviation limits for axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to 32767measurement units

	0	8	2	GRDSX
	0	8	3	GRDSY
	0	8	4	GRDSZ
	0	8	5	GRDS4
	4	2	1	GRDS5

GRDSX, Y, Z, 4 and 5 are grid drifts for axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to ±32767measurement units

The setting is a + (-) value when the reference point is the drift in the + (-) direction.

	0	8	6	LPGMX
	0	8	7	LPGMY
	0	8	8	LPGMZ
	0	8	9	LPGM4
	4	2	2	LPGM5

LPGMX, Y, Z, 4 and 5 are the settings of servo loop gain multiplier for axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings = $2048 \times \frac{E}{L} \times \gamma \times 1000$

E = 7V (motor: 7V/1000rpm)

3.5V(motor: 7V/2000rpm)

L = Mechanical travel in mm or inch equivalent to one rotation of motor

γ= Measurement unit (mm or inch)

Example: Servo motor: 2mm travel each rotation, 7V/1000r/min

Measurement unit: for 1/1000mm

Setting $2048 \times \frac{7}{2} \times \frac{1}{1000} \times 1000 = 7168$

The digits behind the decimal point are rounded off.

	0	9	0	LPGIN

LPGIN The setting of loop gain for position control

Settings 1 to 9999, unit: $0.01s^{-1}$

	0	9	1	JOGF

JOGF The JOG feedrate when the rotary switch is set to 10.

Settings 1 to 150, unit: mm/min, deg/min (output in mm)

1 to 60, unit: 0.1inch/min, 0.1deg/min(output in inch) or 1deg/min(output in inch)

(Note) For the input in inch system, the unit for additional axis is 0.1deg/min or 1deg/min is set

by parameter ROT10(No.306). Also see parameters ADNW(No.318) and JOGFAD(348)

	0	9	2	RPDFX
	0	9	3	RPDFY
	0	9	4	RPDFZ
	0	9	5	RPDF4
	4	2	8	RPDF5

RPDFX, Y, Z, 4 and 5 are the quick speeds of for axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 30 to 15000 unit: mm/min (output in metric system)

30 to 6000 unit: 0.1inch (output in inch system)

	0	9	6	LINTX
	0	9	7	LINTY
	0	9	8	LINTZ
	0	9	9	LINT4
	4	2	9	LINT5

LINTX, Y, Z, 4 and 5 are the time constant (for quick speed) of linear acceleration/deceleration for axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 8 to 4000, unit: msec.

	1	0	0	EXPTX
	1	0	1	EXPTY
	1	0	2	EXPTZ
	1	0	3	EXPT4
	4	3	0	EXPT5

EXPTZ, Y, Z, 4 and 5 are exponential acceleration/deceleration time constant for axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 8 to 4000 unit: m.s

	1	0	5	FEEDT

FEEDT: The exponential acceleration/deceleration time constant for cutting feed
Settings 8 to 4000, unit: m.s.

	1	0	6	FEDMX

FEDMX The upper-limit rate of cutting feed, valid for all axes

(a) For parameter No. 318 (ADNW = 0) (type A)

It is applicable for all axes and sets a tangential speed limit, which shall not be exceeded; otherwise it is subject to clamping.

Settings 6 to 15000, unit: mm/min, output in deg/min metric system

6 to 6000, unit: 0.1inch/min, 0.1deg/min or 1deg/min (output in inch system)

(Note) For output in Inch, the unit for an additional axis is 0.1deg/min or 1deg/min depends on parameter ROT10 (No.306).

(b) For parameter No. 318 (ADNW = 1) (type B)

It is applicable for axes X, Y and Z. the speeds of all axes in linear interpolation and tangential speeds in arc interpolation shall not be exceeded; otherwise they will be restricted.

Settings 6 to 15000 unit: mm/min (output in metric system)

6 to 6000 unit: 0.1inch/min (output in inch system)

(See parameter No. 366 for additional axes)

	1	0	7	EXDEC

EXDEC Speed of external deceleration (common for all axes)

Settings 6 to 15000, unit: mm/min (output in metric system)

6 to 6000, unit: 0.1inch/min(output in inch system)

	1	0	8	FEDFL

FEDFL The lower-limit (FL) rate of exponential acceleration/deceleration for cutting feed

Settings 6 to 15000, unit: mm/min or

6 to 6000 unit 0.1inch/min

The value is generally 0.

	1	0	9	JGFLX

	1	1	0	JGFLY

	1	1	1	JGFLZ

	1	1	2	JGFL4

	4	3	1	JGFL5

JGFLX, Y, Z, 4 and 5 are lower limit (FL) of exponential acceleration/deceleration for continuous JOG feed for axes X, Y and Z as well as the 4^{th} and 5^{th} axes respectively.

Settings 6 to 15000, unit: mm/min.

6 to 6000, unit: 0.1inch/min

	1	1	3	SPDFL

SPDFL The lowest speed (F0) for quick adjustment (common for all axes)

Settings 6 to 15000, unit: mm/min, deg/min(output in metric system).

6 to 6000 unit: 0.1inch/min

0.1deg/min (output in inch system)

6 to 6000 unit: 1deg/min(output in inch system).

(Note) For output in inch system, the unit for an additional axis is 0.1deg/min or 1deg/min depends on parameter ROT10 (306).

	1	1	4	ZRNFL

ZRNFL Low-speed feedrate (FL) for return to the reference point (common for all axes)

Settings 6 to 15000, unit: mm/min, deg/min (output in metric system)

6 to 6000, unit: 0.1inch/min

0.1deg/min or 1deg/min (output in inch system).

(Note) For output in inch system, the unit for an additional axis is 0.1deg/min or 1deg/min depends on parameter ROT10 (No.306).

	1	1	5	BKLX
	1	1	6	BKLY
	1	1	7	BKLZ
	1	1	8	BKL4
	4	3	2	BKL5

BKLX, Y, Z, 4 and 5 are lower limit (FL) of reverse clearances for axes X, Y and Z as well as the 4^{th} and 5^{th} axes respectively.

Settings 0 to 255unit 0.001mm(output in metric system)

0 to 255unit 0.0001inch(output in inch system)

	1	1	9	SPDLC

SPDLC Spindle speed offset compensation, i.e. the compensation for setting the zero offset of spindle speed instruction voltage (for S analog output A/B)

Settings 0 to ±8191unit VELO.

	1	2	1	TLCNEG

TLCNEG Tool life management negligence number

Settings 1 to 255

	1	2	4	DRFTX
	1	2	5	DRFTY
	1	2	6	DRFTZ
	1	2	7	DRFT4
	4	2	3	DRFT5

DRFTX, Y, Z, 4 and 5 are the compensation of the drift occurred inside the servo ring of axes X, Y and Z as well as the 4^{th} and 5^{th} axes respectively.

Settings 0 to ±5000 unit: VELO

The value automatically changes after the parameter setting (007-ADFT) of automatic drift compensation.

	1	2	8	PRAZX
	1	2	9	PRAZY
	1	3	0	PRAZZ
	1	3	1	PRAZ4

	4	2	4	PRAZ5

PHAZX, Y, Z, 4 and 5 are the servo phase deviation of axes X, Y and Z as well as the 4^{th} and 5^{th} axes respectively. The value corresponding to the signal phase fed back from phase detector is automatically set (for rotary transformer and inductosyn).

Settings 0 to 500

	1	3	2	GRLMAX

GRLMAX The setting of maximum number of spindle revolutions for low-speed gear (S12-digit outputs A, for S analog output A)

The setting of number of spindle revolutions when speed instruction voltage is 10V.

Settings 1 to 9999 unit: r/min

	1	3	3	GRHMAX

GRHMAX The setting of maximum number of spindle revolutions for high-speed gear (S12-digit outputs A, for S analog output A)

The setting of number of spindle revolutions when speed instruction voltage is 10V.

Settings 1 to 9999 unit r/min

	1	3	4	GRHMIN

GRHMIN The setting of lower limit of spindle revolutions for high-speed gear (S12-digit outputs A, for S analog output A)

Settings 1 to 9999 unit r/min.

(Note) It becomes a low-speed gear when a value identical with the setting is instructed.

	1	3	5	SPDMIN

SPDMIN The setting of lower limit of spindle motor output (S12-digit outputs A/B, for S analog output A/B)

$$\text{Settings} = \frac{\text{Lower limit of spindle motor}}{\text{maximum rotations of spindle motor}} \times 4095$$

Settings 1 to 4095

	1	3	6	SPDMAX

SPDMAX The setting of upper limit of spindle motor output (S12-digit outputs A/B, for S analog output A/B)

$$\text{Settings} = \frac{\text{Upper limit of spindle motor}}{\text{maximum rotations of spindle motor}} \times 4095$$

Settings 1 to 4095

	1	4	0	PSANGN

PSANGN Setting of the gain calling data of S analog output A/B

Setting range: 700 to 1250

Standard setting: 1000

[Adjusting procedures]

(1) Set it to standard setting “1000”.

(2) Specify the maximum value of S analog quantity (10V).

(3) Measure the output voltage.

(4) Set PSANGN according to the following formula.

$$\frac{10.0}{\text{Measured voltage } (V)} \times 1000 = \text{Setting value}$$

(5) Set the parameter and then instruct the maximum (10V) confirming output voltage of S analog quantity as 10V.

	1	4	1	TIME1

TIME1 Set the preset quantity of service time.

It is also possible to preset with setting.

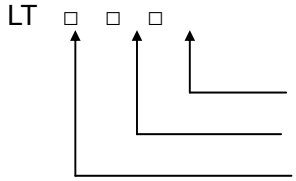
Settings 0 to 32767 unit: 0.1hr

	1	4	2	TIME2

TIME2 Set the preset quantity of service time.

Settings 0 to 99999999 unit: 0.1hr

	1	4	3	LT1X1
	1	4	4	LT1Y1
	1	4	5	LT1Z1
	1	4	6	LT141
	4	3	3	LTI51
	1	4	7	LT1X2
	1	4	8	LT1Y2
	1	4	9	LTIZ2
	1	5	0	LT142
	4	3	4	LT152
	1	5	1	LT2X1
	1	5	2	LT2Y1
	1	5	3	LT2Z1
	1	5	5	LT2X2
	1	5	6	LT2Y2
	1	5	7	LT2Z2



1

Set the above travel limit. 2

Settings 0 to ±99999999 unit: 0.001mm (output in metric system)

0 to ±99999999 unit: 0.0001inch(output in inch system)

151 to 157 can also be set with setting values.

	1	5	9	REF2X
	1	6	0	REF2Y
	1	6	1	REF2Z
	1	6	2	REF24
	4	3	5	REF25

REF2X, Y, Z, 4 and 5 are the distance from the 2nd reference points to the 1st one of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to ±99999999, unit: 0.001mm(output in metric system).

0 to ±99999999, unit: 0.0001inch(output in inch system).

0 to ±99999999, unit: 0.001°(axis of rotation).

	1	6	3	PECINTX
	1	6	4	PECINTY
	1	6	5	PECINTZ
	1	6	6	PECINT4
	4	3	6	PECINT5

PECINTX to 5 are the settings of pitch error compensation spacing of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 8000 to 20000000, unit: 0.001mm (output in metric system).

4000 to 20000000, unit: 0.0001inch (output in inch system).

6000 Unit: 0.001°(axis of rotation)

(Note) The setting to 0 does not compensate.

	1	6	7	ATCLZV

ATCLZV The setting of travel limit of axis Z in negative direction

Settings 0 to ±99999999, unit: 0.001mm (output in metric system).

0 to ±99999999, unit: 0.0001inch (output in inch system).

	1	6	8	Password

MASKA Store a password before using key lock for a program.

Settings 1 to 99999999

	1	7	1	FIDF1
	1	7	2	FIDF2
	1	7	3	FIDF3
	1	7	4	FIDF4
	1	7	5	FIDF5
	1	7	6	FIDF6
	1	7	7	FIDF7
	1	7	8	FIDF8
	1	7	9	FIDF9

FIDF1, 2, 3, 4, 5, 6, 7, 8 and 9 are the feedrates corresponding to F1 instructions F1 to F9.

Settings 0 to 15000 unit: 0.1mm/min (output in metric system).

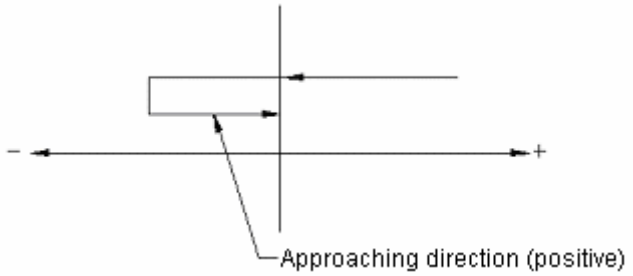
0 to 6000, unit: 0.01inch/min (output in inch system).

The setting with set values is also possible.

In addition: For F1 instructions, the parameter value changes when feedrate is changed with a manual pulse generator.

	3	0	5	FL4	FLZ	FLY	FLX	G604	G60Z	G60Y	G60X
				7	6	5	4	3	2	1	0

G60X, Y, Z and 4 are the approaching directions of the single-directional positioning of axes X, Y and Z and the 4th axis.



1. Approaching direction is negative.

0. Approaching direction is positive.

For FLX, Y, Z, 4, NC/TC, whether the axes X, Y and Z and the 4th axis are parallel to the 5th axis (NC/TC)

1. parallel

0. not parallel

	3	0	6	SKPF	CHR		SFRV	NEOP	ROT10	TMCR	SALM
				7	6	5	4	3	2	1	0

SKPF 1. For skip instruction (G31), the feedrate becomes the FL speed (No.342) set by parameter.

0. For skip instruction (031), feedrate is specified by F code.

CHR 1. For the look-at-me function of handwheel, the feedrate is limited to quick speed.

0. Not limited to quick speed.

SFRV 1. In G84 and G74, SRV can be used to change the polarity of analog voltage.

0. Cannot change

(Note) The parameter is valid only when the parameter NO.010 Bit7 TCW is “1”.

NEOP 1. When storing programs in memory, M02, M30 and M99 are not used for the end of program.

0. May be used for the end of program.

ROT10 1. For output in inch system, the unit of feedrate parameters(No.091, 106, 113 and 114) is 1deg/min.

0. The unit is 0.1deg/min.

(The parameter is valid only when the additional axis is an axis of rotation.)

TMCR 0. T code cannot be used for calling

1. T code is used for calling a user macro.

SALM 1. Alarm is given when the S code instructed in S4 binary 12-digit output A/analog output A goes beyond the lower limit or upper limit of the value output to the spindle.

0. No give any alarm but be restrained to the lower limit or upper limit.

Refer to parameters No.135 and 136.

	3	0	7		EX4NG	SFOB	SCDB	GRST		TLCD	
				7	6	5	4	3	2	1	0

EX4NG 1. Additional-axis neglecting signal 4NG is valid.

0. Additional-axis neglecting signal 4NG is invalid.

Whether SFOB outputs SF in S12-digit output B or S analogy output B:

1. Not output SF.

0. Output SF.

SCDB 1. The last 2 digits of S4-digit is output to B21 through B38 for S12-digit output B or analogy output B. If B3-digit function is required, the digit can not be set to 1.

0. Not output

GRSR 1. All executive data in all groups are cleared during the input of tool change reset signal.

0. Only the executive data that tool life times out are cleared during the input of tool change reset signal.

TLCD 1. Tool length compensation is added to an instructed axis.

0. Tool length compensation is usually added to axis Z.

	3	0	8	DIOM	MSFT	LGCM		RSTB		CFMF	
				7	6	5	4	3	2	1	0

DIOM 1. It is possible to read and write DI and DO using macro variables.

0. Impossible.

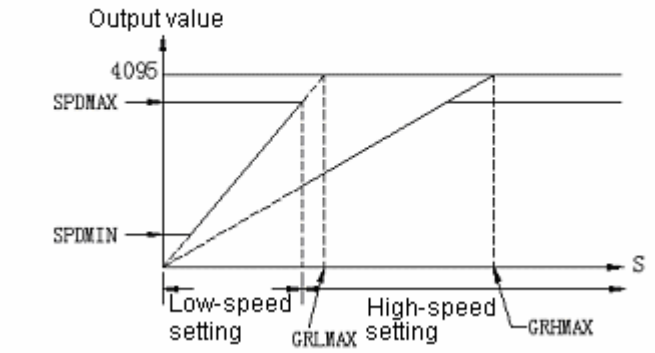
MSFT 1. If user macro selection is provided, SHIFT key is active when typing in through MDI.

0. SHIFT key is active when typing in through MDI.

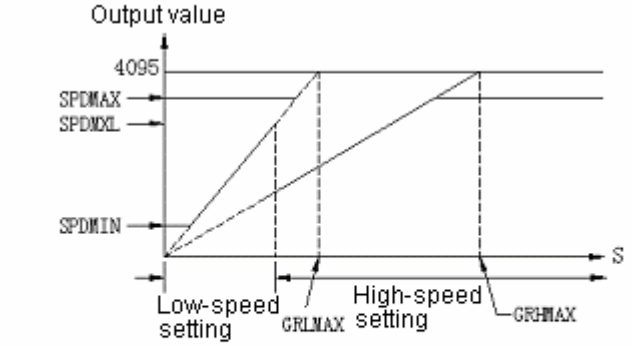
LGCM 1. The number of rotations for switching between low-speed and high-speed gears depends on the value of parameter SPDMXL(NO·365) (type A).

0. The number of rotations for switching is the maximum value at low speed (type A).

Type A (when parameter LGCM=0)

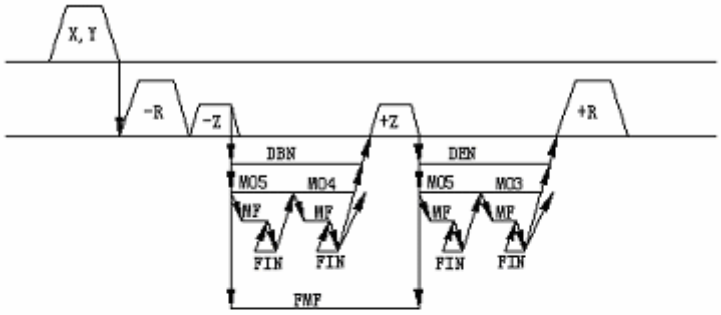


Type B (when parameter LGCM=1)

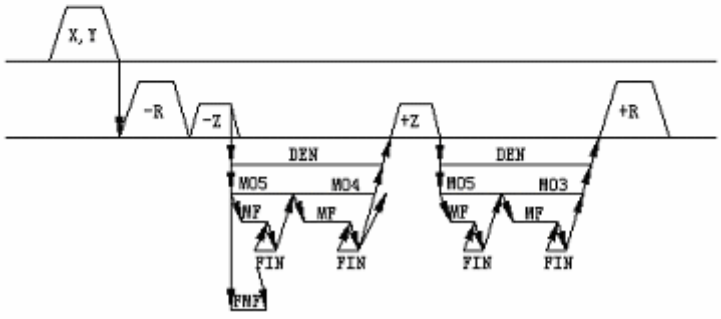


GRLMAX (Parameter NO. 132)
GRHMAX (Parameter NO. 133)
SPDMIN (Parameter NO. 135)
SPDMAX (Parameter NO. 136)
SPDMXL (Parameter NO. 365)

Time sequence A (when CFMF=0)



Time sequence B (when CFMF=1)



Note: The parameter is valid for S12-digit output A and S analog output A.

RSTB 1. In-reset signal is not output when the system is reset by with emergency stop,

external reset, reset and rewinding.

0. Output in-reset signal.

CFMF 1. For G84 and G74 in the fixed cycle II, output signal FMF is switched OFF by the FIN of M05.

0. For G84 and G74 in the fixed cycle II, output signal FMF is output until R-point plane.

	3	0	9	TLSK	GST2	GST1	LCTM	AP4	APZ	APY	APX
				7	6	5	4	3	2	1	0

TLCK 1. Input group number during tool skip.

0. Not input group number.

GST1, GST2

GST2	GST1	Group No.
0	0	1 to 16
0	1	1 to 32
1	0	1 to 64
1	1	1 to 128

LCTM 1. Specify tool life in time.

0. Specify tool life in number of cycles.

APX, Y, Z and 4 are the settings of availability of automatic coordinate setting of axes X, Y and Z and the 4th axis respectively.

1. Enable automatic coordinate setting

0. Disable.

See parameters No.375 ～ 382.

	3	1	0	NFED1		RSCR1	STP21	RAD1
	3	1	1	NFED2		RSCR2	STP22	RAD2
	3	1	2	NFED3		RSCR3	STP23	RAD3
	3	1	3	NFED4		RSCR4	STP24	RAD4

When NFED1, 2, 3 and 4 use I/Os 1, 2, 3 and 4 respectively, whether the blank space between the first and last guide holes and program is output or not:

1. Not output guide space

0. Output.

RSCB1, 2, 3, 4

When RSCB 1, 2, 3 and 4 use I/Os 1, 2, 3 and 4 respectively, whether the control codes (DC1 to DC4) are used or not:

1. Not use the control codes.

0. Use the control codes.

When STP21, 2, 3 and 4 use I/Os 1, 2, 3 and 4 respectively, the number of stop bit(s) is set to 2 or 1:

1. 2 stop bits

0. 1 stop bit

RAD1, 2, 3 and 4 are the baud rate settings for I/Os 1, 2, 3 and 4.

(Note) If ROBOT interface selection is provided, the baud rate for the data transmission between NC and ROBOT is set to BAD4.

Baud rate	RAD1, 2, 3, 4			
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1
9600	1	0	1	0

(Note) See parameters No.340 and 341.

	3	1	4

IM15	MINT	IFIX	IRND	H4	HZ	HY	HX
7	6	5	4	3	2	1	0

IM15 1. For the instructions for axis B, its rotating direction is positive despite that G90/G91 mode must be regarded as an absolute instruction.

In addition: M15 is instructed to rotate in negative direction.

0. The instruction for axis B is an absolute/incremental instruction depending on G90/G91 and its rotating direction is positive or negative. M15 has no special meaning.

MINT 1. Start to execute interrupt program (user macro interrupt type II) at the end of the execution of the current block.

0. Immediately execute the interrupt program (user macro interrupt type I)

IFIX 1. Give P/S alarm for the instructions other than integral multiple of the indexing angle of indexing workbench (parameter No.060 shall be also be set).

0. The instructions for axis B may instruct the instructions independent of the minimum angle of the indexing workbench.

IRND 1. It is possible to round the absolute coordinates of axis B to 360.

0. It is impossible to round them to 360.

HX, Y, Z and 4 are the setting of interrupts of the handwheels of axes X, Y and Z as well as the 4^{th} axis.

1. Enable.

0. Disable.

	3	1	5	PRT	BLOW	BDEG	IDXB	SSCR	SSCA2	SSCA1	SSCA0
				7	6	5	4	3	2	1	0

PRT 1. Leading zero outputs nothing while DPRNT instruction is used for data input.

0. Leading zero outputs blank spaces while DPRNT instruction is used for data input.

SLOW 1. The parameter setting of the clamping value for the minimum number of spindle rotations in the constant control of surface speed is applicable for all gears. (No. 347)

0. Set for all gears respectively (No. 343, 344, 345 and 346)

BDEG 1. Input unit: 0. 001°(B1=0．001°).

0. Input unit of axis B: 1°(B1=1°).

IDXB 1. Indexing sequence B of indexing workbench.

0. Indexing sequence A of indexing workbench.

SSCR 1. In the blocks of rapid feed, surface speed is calculated in accordance with the coordinates of the end point of the current block.

0. In the blocks of rapid feed, surface speed is calculated in accordance with the actual position of the tool.

SSCA2, SSCA1 and SSCA0 are settings for the axis as the calculation basis in the constant control of surface speed.

SSCA2	SSCA1	SSCA0	Axis
0	0	0	X
0	0	1	Y
0	1	0	Z
0	1	1	4
1	0	0	5

	3	1	6	CDSCG		ACMR	DSCG5	DSCG4	DSCGZ	DSCGY	DSCGX
				7	6	5	4	3	2	1	0

CDSCG 1. Not perform frequency detection of DSCG feedback (rotary transformer and inductosyn).

0. Perform detection

(The parameter is always preset to 0 after original adjustment.)

ACMR 1. It is possible to make special CMR setting (disabled).

0. It is impossible to set special CMR.

DSCGX, Y, Z, 4 and 5 are the position detection system type settings of axes X, Y and Z as well as the 4th and 5th axes respectively.

1. Position detection system is a rotary transformer or inductosyn.

0. Position detection system is a pulse coder.

(Note) Combination of pulse coder and rotary transformer or inductosyn in axes X, Y and Z is not allowed. The mixture of the 4th and 5th axes is also not allowed.

	3	1	7	UM#8	UM#7	UM#6	UM#5	UM#4	UM#3	UM#2	UM#1
				7	6	5	4	3	2	1	0

UM#1 to 8 For EIA code, store and use the codes corresponding to “#” used in macro.

Example: UM#8 to UM#1=01001001

The codes with holes on passage 1, 4 and 7 are regarded as the "#" of EIA. The used address codes cannot be set.

UM#8 to UM#1=00000000 indicates that "#" will not be used.

	3	1	8

PRG9	MSC9	MPD9			NSRH	RSTL	ADNW
7	6	5	4	3	2	1	0

PRG9 1. The programs of numbers 9000 to 9899 cannot be edited.

0. The programs of numbers 9000 to 9899 can be edited.

MSC9 1. When executing the programs of numbers 9000 to 9899, single block stops while executing a macroinstruction in user macro if it is single block mode.

0. Not execute.

MPD9 1. The contents of program are not displayed when executing the programs of numbers 9000 to 9899.

0. The contents of program are displayed when executing the programs of numbers 9000 to 9899.

MSRH 1. "OP" signal is not output during sequence number search.

0. "OP" signal is output during sequence number search.

RSTL 1. STL signal is not output when using CYCLE START to store messages in memory in EDIT mode.

0. STL signal is output.

ADNW 1. Feedrate is of type B.

0. Feedrate is of type A.

[Type B]

(1) JOG feedrate

The JOG feedrate of additional axis (axis of rotation) is set by parameter No.348. When the additional axis is interlocked with another axis or it is a linear axis (for parameter No. 11 BIT0 ANLN=1), however, the JOG feedrate of the additional axis is identical with other axes (parameter No. 091 JOGF).

(2) Upper-limit rate of cutting feed

When instructing linear interpolation (G01), the value that speeds of all axes exceed parameter setting by is clamped.

The clamped value is set for axes X, Y and Z and additional axis individually. For arc interpolation, the value that tangential speed exceeds parameter setting by is clamped.

[Type A]

(1) JOG feedrate

The feedrate of additional axis is set along with other axes in parameter No.091.

(2) Upper-limit rate of cutting feed

That the tangential speeds of all axes exceed parameter value is limited by parameter.

	3	1	9

PRG8	MCS8	MPD8					MCS7
7	6	5	4	3	2	1	0

It is also possible to set with set values.

PRG8 1. It is impossible to edit the programs of program numbers 8000 to 8999

0. Possible

MCS8 1. If the programs of program numbers 8000 to 8999 are executed in SINGLE

mode, the execution of the macroinstruction of a user macro stops at a single block.

0. Not stop at a single block.

MPD8 1. The contents of program is not displayed when executing the programs of program numbers 8000 to 8999.

0. The contents of program is displayed

MCS7 1. If the programs of program numbers 0001 to 1999 are executed in SINGLE mode, the execution of macro stops at a single block.

0. Not stop at a single block.

	3	2	0	UMMCD1
	3	2	1	UMMCD2
	3	2	2	UMMCD3

UMMCD 1, 2, 3 Up to 3 M codes for calling user macro are set.

Settings 01 to 91.

(User macro cannot be called with M00. The setting of 00 is equivalent to no setting.)

	3	2	3	UMGCD0
	3	2	4	UMGCD1
	3	2	5	UMGCD2
	3	2	6	UMGCD3
	3	2	7	UMGCD4
	3	2	8	UMGCD5
	3	2	9	UMGCD6
	3	3	0	UMGCD7
	3	3	1	UMGCD8
	3	3	2	UMGCD9

UMGCD0．1……9
Up to 10 G codes for calling user macro are set.

Settings 001 to 255.

(User macro cannot be called with G00. The setting of 00 is equivalent to no setting.)

	3	3	3	AOVMDR

AOVMDR The minimum reduction ratio of interior arc cutting rate
Range: 1 to 100%, standard setting: 1

	3	3	4	AOVOR

AOVOR The minimum reduction ratio of automatic adjustment for interior turning angle section

Range: 1 to 100%, standard setting: 50

Set the adjusting value of inner turning angle.

	3	3	5	AOVTR

AOVTH Interior reference angle for automatic adjustment for interior turning angle.

Range: 1 to 179°, standard setting: 91°

	3	3	6	POSTNX
	3	3	7	POSTNY
	3	3	8	POSTNZ
	3	3	9	POSTN4
	4	1	7	POSTN5

POSTN X, Y, Z, 4 and 5 are the approaching amount of negative positioning of axes X, Y and Z as well as the 4^{th} and 5^{th} axes respectively.

Settings 0 to 255; unit: 0.01mm (output in mm).

0 to 255; unit: 0.001 inch (output in inch).

	3	4	0	IDVICE
	3	4	1	ODVICE

IDVICE: Select an input device (INPUT DEVICE2=1(RS232) interface for setting; this setting is valid) for storing programs in memory.

ODVICE: Select an output device for data output.

Settings	I/O
0	Paper tape reader for input and FACIT and PUNCHER for output
1	ASR33/ASR43 is used to set baud rate and other data (No.310) for both input and output.
2	Reader/puncher interface for I/O. Set baud rate and other data in parameter No.311.
3	Reader/puncher interface for I/O. Set baud rate and other data in parameter No.312.
4	Reader/puncher interface for I/O. Set baud rate and other data in parameter No.313.

	3	4	2	PSKPFL

PSKPFL FL speed of skipping cutting (common for all axes)

Settings 6 to 150000 unit: 1mm/min (output in metric system).

6 to 6000 unit: 0.1 inch/min (output in inch system).

	3	4	3	GRMIN1
	3	4	4	GRMIN2
	3	4	5	GRMIN3
	3	4	6	GRMIN4

GRMIN1 to GRMIN4 Minimum number of spindle revolutions in surface speed constant control (G96) mode

Settings 0 to 9999 Unit: RPM.

Valid only when parameter No. 3/5 - SLOW=0

	3	4	7	LOWSP

LOWSP Minimum number of spindle revolutions in surface speed constant control (G96) mode (for constant control selection)

Settings 0 to 9999 unit RPM

Valid only when parameter No. 315 - SLOW=1

	3	4	8	JOGFAD

The JOG feedrate when the rotary switch is set to 10 with the existence of an additional axis (axis of rotation) (type B).

Settings 1 to 150 unit: deg/min

See parameter No.091 (JOGF)

(Note) Refer to parameter No.318 ADNW for type B.

	3	5	5	AOVLE

AOVLE The decelerating distance Le at the end point of the automatic adjustment of interior turning angle section

Range: 0 to 3999 unit: 0.1mm(input in metric system).

Unit 0.01 inch(input in inch system).

The parameter may be operated and set with set values.

	3	5	6	AOVLS

AOVLS The decelerating distance Ls at the end point of the automatic adjustment of interior turning angle section

Range 0 to 3999; unit: 0.1mm (input in metric system).

Unit: 0.01 inch (input in inch system).

The parameter may be operated and set with set values.

	3	5	7	EXOFSX

	3	5	8	EXOFSY

	3	5	9	EXOFSZ

	3	6	0	EXOFS4

	4	4	2	EXOF5

EXOFS X, Y, Z, 4 and 5 are external workpiece origin offset of axes X, Y and Z as well as the 4^{th} and 5^{th} axes.

Settings 0 to ±7999 unit: 0.001mm (input in metric system).

0 to ±7999 unit: 0.0001 inch(input in inch system).

Typically the parameter is automatically set by the input on the machine side.

(External data input function).

	3	6	1	PGMAX1

	3	6	2	PGMAX2
	3	6	3	PGMAX3
	3	6	4	PGMAX4

PGMAX1, 2, 3 and 4 select the maximum set numbers of rotations of gears 1, 2, 3 and 4 (For S12-digit output B and S analog output B).

Set the number of spindle rotations at 10V speed instruction voltage.

Settings 1 to 9999rpm.

	3	6	5	SPDMXL

SPDMXL Settings = $\frac{\text{Permissible maximum number of rotations of spindle motor}}{\text{Maximum number of rotations of spindle motor}} \times 4095$

Setting range: 0 to 4095

Valid only when parameter NO. 308－5 LGCM =1

	3	6	6	FEDMXAD

FEDMXAD Upper-limit rate of the cutting feed of additional axis

Valid only when parameter No.318 (ADNW=1)

Settings 6 to 15000; unit deg/min (axis of rotation)

6 to 15000; unit: mm/min (output in metric system)

6 to 6000; unit: inch/min (output in inch system)

	3	6	7	REF3X
	3	6	8	REF3Y
	3	6	9	REF3Z
	3	7	0	REF34
	4	3	8	REF35

REF3X, Y, Z, 4 and 5 are the distances from the 3rd reference point to the 1st reference point of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to ±99999999 unit: 0.001mm (output in metric system)

0 to ±99999999 unit: 0.0001inch (output in inch system)

	3	7	1	REF4X
	3	7	2	REF4Y
	3	7	3	REF4Z
	3	7	4	REF44
				REF45

	4	3	9

REF4X, Y, Z, 4 and 5 are the distances from the 4th reference point to the 1st reference point of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to ±99999999 unit: 0.001mm (output in metric system)

0 to ±99999999 unit: 0.0001inch (output in inch system)

	3	7	5	PPRTMX

	3	7	6	PPRTMY

	3	7	7	PPRTMZ

	3	7	8	PPRTM4

	4	4	0	PPRTM5

PPRTMX, Y, Z, 4 and 5 are settings of automatic coordinate system input in metric system of axes X, Y and Z as well as the 4th and 5th axes respectively. The distance from the origin of coordinate system to the first reference point is set in metric system.

Settings 0 to 99999999 unit: 0.001mm

If Inch/metric switching option is provided, parameter No.379 to 382 and 411 shall also be set.

It is only valid for the axes that are set active in the automatic coordinate system set in parameter No.309.

	3	7	9	PPRTIX

	3	8	0	PPRTIY

	3	8	1	PPRTIZ

	3	8	2	PPRTI4

	4	4	1	PPRTI5

PPRTIX, Y, Z, 4 and 5 are settings of automatic coordinate system input in inch system of axes X, Y and Z as well as the 4th and 5th axes respectively.

The distance from the origin of coordinate system to the first reference point is set in inch system.

When settings 0 to 99999999 (unit 0. 0001inch) are provided with Inch/metric switching option, parameters No. 375 to 378 and 440 shall be set. This is only valid for the axes that are set active in the automatic coordinate system set in parameter No.309.

	3	8	3	ZOFSIX

	3	8	4	ZOFSIY

	3	8	5	ZOFSIZ

	3	8	6	ZOFSI4

	4	4	3	ZOFSI5

ZOFSIX, Y, Z, 4 and 5 are the first workpiece origin offsets (G54) of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to 99999999 unit: 0.001mm (input in metric system)

0 to 99999999 unit: 0.0001inch (input in inch system)

The OST function key is usually selected for input.

	3	8	7	ZOFS2 X
	3	8	8	ZOFS2 Y
	3	8	9	ZOFS2 Z
	3	9	0	ZOFS2 4
	4	4	4	ZOFS2 5

ZOFS2X, Y, Z, 4 and 5 are the 2nd workpiece origin offsets (G55) of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to 99999999 unit: 0.001mm(input in metric system)

0 to 99999999 unit: 0.000inch(input in inch system)

The OST function key is usually selected for input.

	3	9	1	ZOFS3 X
	3	9	2	ZOFS3 Y
	3	9	3	ZOFS3 Z
	3	9	4	ZOFS3 4
	4	4	5	ZOFS3 5

ZOFS3X, Y, Z, 4 and 5 are the 3rd workpiece origin offsets (G56) of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to 99999999 unit 0. 001mm(input in metric system).

0 to 99999999 unit 0. 0001inch(input in inch system).

The OST function key is usually selected for input.

	3	9	5	ZOFS4 X
	3	9	6	ZOFS4 Y
	3	9	7	ZOFS4 Z
	3	9	8	ZOFS4 4
	4	4	6	ZOFS4 5

ZOFS4X, Y, Z, 4 and 5 are the 4th workpiece origin offsets (G57) of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to 99999999 unit 0.001mm (input in metric system).

0 to 99999999 unit 0.0001inch(input in inch system).

The OST function key is usually selected for input.

	3	9	9	ZOFS5 X

	4	0	0	ZOFS5 Y
	4	0	1	ZOFS5 Z
	4	0	2	ZOFS5 4
	4	4	7	ZOFS5 5

ZOFS5X, Y, Z, 4 and 5 are the 4th workpiece origin offsets (G57) of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to 99999999 unit 0.001mm (input in metric system).

0 to 99999999 unit 0.0001inch(input in inch system).

The OST function key is usually selected for input.

	4	0	3	ZOFS6 X
	4	0	4	ZOFS6 Y
	4	0	5	ZOFS6 Z
	4	0	6	ZOFS6 4
	4	4	8	ZOFS6 5

ZOFS6X, Y, Z, 4 and 5 are the 5th workpiece origin offsets (G57) of axes X, Y and Z as well as the 4th and 5th axes respectively.

Settings 0 to 99999999 unit 0.001mm (input in metric system).

0 to 99999999 unit 0.0001inch(input in inch system).

The OST function key is usually selected for input.

	4	0	7	SCRATE

SCRATE Scaling rate

Settings: 0 to 99999999; unit: 0.001 time.

It is a value whose P has not been instructed in the blocks of G51.

	4	0	8	LOCK/UNLOCK

Typing will not be locked when a value identical with that of No.168 is input. It is locked in the event of input of different values.

	4	1	1					ZGM5	ZM5
				7	6	5	4	3	2	1	0

ZGM5 The return-to-reference point mode of the 5th axis

1. Magnetic switch mode
0. Grid mode

ZM5 1. The returning direction to reference point and the initial direction of clearance of the 5th axis are negative.

0. The returning direction to reference point and the initial direction of clearance of the 5th axis are positive.

	4	1	2		ADW52	ADW51	ADW50	AD5B	AD5A	AD4B	AD4A
				7	6	5	4	3	2	1	0

The name selection of the 5th axis when ADW52, 1 and 0 have the 5th axis.

ADW52	ADW51	ADW50	Letter
0	0	0	A
0	0	1	B
0	1	0	C
0	1	1	U
1	0	0	V
1	0	1	W

AD4A, AD4B, AD5A, AD5B
Setting of which axis the 4th and 5th axes parallel

The 5th axis		The 4th axis		The basic axis that parallels the 4th and 5th axes
AD5B	AD5A	AD4B	AD4A	
0	0	0	0	Axis X
0	1	0	1	Axis Y
1	0	1	0	Axis Z
1	1	1	1	Which axis it does not parallel

	4	1	3

CLCL		WNSAA	
7	6	5	4 3 2 1 0

CLCL 1. Local coordinate system is cleared after the JOG return to reference point

0. Not cleared.

WNSAA 1. External workpiece number search function is used to search the programs of a program number whose first digits are 00 and last two digits are identical with the search number.

0. External workpiece number search function is used to search the programs of a program number whose last two digits are identical with the search number.

	6	3	3

ROTR	
7	6 5 4 3 2 1 0

ROTR 1. The R value of angle instruction is absolute or incremental depends on G90/G91 mode.

0. The R value of angle instruction is absolute.

	7	1	6

RTANGL
7 6 5 4 3 2 1 0

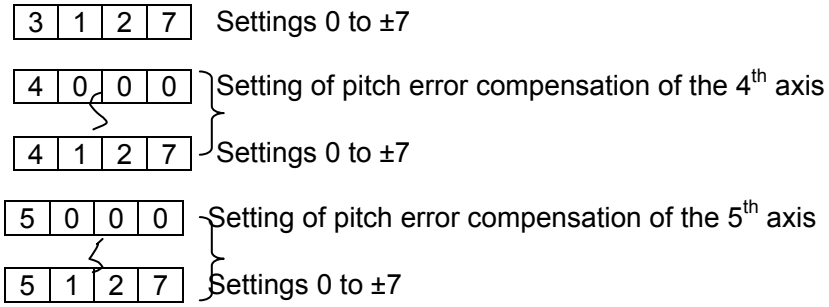
(The set value may also be set).

RTANGL The angle that the coordinate system rotates by

Settings: 0 to 360000; unit: 0. 001deg.

The value is valid when it is not specified in the blocks of G68.

1	0	0	0	Setting of pitch error compensation of axis X
1	1	2	7	Settings 0 to ±7
2	0	0	0	Setting of pitch error compensation of axis Y
2	1	2	7	Settings 0 to ±7
3	0	0	0	Setting of pitch error compensation of axis Z


3127
Settings 0 to ±7
4000
Setting of pitch error compensation of the 4th axis
4127
Settings 0 to ±7
5000
Setting of pitch error compensation of the 5th axis
5127
Settings 0 to ±7

The items set and displayed using set values.

Data No.	Item
000	The settings related to I/O (DNC, RMT, INCH, ISO, TVON, REV4, REVY and REVX)
057*	Machining time (unit: hr) (TMHOR)
058*	Machining time (unit: min) (TMMIN)
059*	Machining time (unit: sec) (TMSEC)
067*	The returning amount in fixed cycle G83 (high-speed depth drilling cycle) (CYCR)
068*	The cutting origin in fixed cycle G83 (high-speed depth drilling cycle) (CYCD)
141*	Machining time (TIME1)
151*	Storage type travel limit 2, X value of the 1st acme
152*	Storage type travel limit 2, Y value of the 1st acme
153*	Storage type travel limit 2, Z value of the 1st acme
155*	Storage type travel limit 2, X value of the 2nd acme
156*	Storage type travel limit 2, Y value of the 2nd acme
157*	Storage type travel limit 2, Z value of the 2nd acme
180*	The program number of the program whose execution is stopped
319*	All settings (PRG8 and MSBL).
340*	Select an input device for data storage (IDVICE)
341*	Select an output device for data output (ODVICE)
355*	The decelerating distance at the end point of a block (automatic adjustment of turning angle)
356*	The decelerating distance at the starting point of a block (automatic adjustment of turning angle)
407*	Scaling factor

- Select address SET.
- The data numbers other than other listed in the table above are displayed as blank.
- The data number with a “*” can be set with address PARAM in the same data number.
- See the parameter explanations regarding the same data numbers for the contents.

Appendix 6 Alarms List

No.	Descriptions	Remarks
000	N O power supply must be disconnected once and reconnected after parameter setting (parameter No. 012 to 018, 027 to 034, 082 to 090 and 124 to 131, 316).	
001	TH alarm (A character with incorrect parity is input in valid information area)	
002	TV alarm (the number of characters in a block is odd). ON is possible only when TV detection is set.	
003	A data with impermissible number of digits is input (see the item of “Maximum instruction value”)	
004	Numeral, character (-) or decimal point is entered without address at the beginning of a block (refer to the descriptions for P/S alarm in the items III, 10 and 11 when it is provided with user macro option).	
005	There is not data but address or EOB code following an address.	
006	Sign “-“ is input incorrectly (“-“ is entered behind an address that allows no negative sign, or two or more “-“ signs are entered.)	
007	Decimal point is entered incorrectly (“·“ is entered behind an address that allows no negative sign, or two or more “·“symbols are entered.)	
008	Device setting is entered incorrectly.	
009	A prohibited character is entered in the valid information area. (E)	
010	A prohibited G code is used (the alarm is also given when an instruction similar to the unselected G code is instructed).	
011	Feedrate is not instructed or instructed improperly during cutting feed.	
014	Synchronous feed is instructed when thread cutting/synchronous feed option is not provided.	
015	Permissible number of concurrently controllable axes is exceeded.	
017	The move instruction of additional axis is instructed without additional control option.	
018	Concurrent motion of additional axis and other axes is instructed without additional control select function for additional axis.	
021	An axis beyond plane assignments (G17, G18 and G19) is instructed in arc instruction.	
022	3R is instructed without radius R instruction option in arc instruction.	
023	R is instructed as 0 or negative when radius R instruction is used in arc instruction.	

027	For an axis, tool length compensation is applied without cancellation of the foregoing tool length compensation.	
028	An instruction of more than 2 axes is instructed for the axis in the same direction in an arc instruction.	
029	Compensation is modified when the stored offset is of more than 6 digits.	
030	The compensation number instructed with the D and H codes for tool radius compensation, tool length compensation and tool position compensation is excessive.	
031	The P value of an instructed offset number is excessive or P is not instructed in the input of offset (using G10 or user macro input command).	
032	The offset instructed by P is excessive in the input of offset (using G10 or user macro input command).	
033	Intersection point cannot be determined using the intersection-point method of calculation for tool radius compensation C, or intersection point calculation of a turning angle less than 90° is instructed in tool radius compensation B.	
034	Compensation “start” or “cancel” is executed in G02/03 mode during tool radius compensation.	
035	Skipping cutting (G31) is instructed in tool radius compensation.	
036	G45 to G48 (tool offset) are instructed in tool radius compensation.	
037	Switching between compensation planes (G17, G18 and G19) is conducted in tool radius compensation.	
038	Overcutting is caused as the radius of arc is 0 at its origin and end point in tool radius compensation.	
041	Overcutting is caused in tool radius compensation.	
044	G27 through G30 and ATC cycle (M06) are instructed in fixed cycle mode.	
045	ATC cycle (M06) is instructed in a device without return-to-reference-point selection function.	
046	An instruction other than P2, P3 and P4 is instructed in the 2^{nd}, 3^{rd} and 4^{th} return-to-reference-point instructions.	
047	G27 to G30 are instructed for an axis without reference point.	
048	G30 is instructed without returning to reference point after switching on or emergency stop. When a storage type travel limit option is provided, a move instruction is executed without returning to reference point after switching on or emergency stop.	
058	An instruction beyond the maximum or minimum number of spindle rotations is instructed in S4-digit binary 12-bit/analog output A.	
059	The block of the selected workpiece number is not found (external workpiece number selection A function).	
060	The specified sequence number is not found during sequence number search or restart of program.	
065	A value of a scaling rate beyond 1 to 99999 is instructed.	
066	After the application of scaling, traveling amount, coordinates and arc radius exceed the maximum instruction value.	

067	G51 (scaling ON) is instructed in tool radius compensation.	
070	The capacity of memory is inadequate.	
071	The address searched is not found.	
072	The number of stored programs exceeds 95 or 191.	
073	A stored program number is used.	
074	The program number is beyond 1 to 9999.	
075	Neither program number nor sequence number is entered in the first paragraph of the program.	
076	Address P is not instructed in the blocks containing M98, G65 and G66.	
077	A subprogram is called for 3 times (or 5 times with a user macro option).	
078	The program number (the program number instructed when G, M and T calling are used) specified by address P, sequence number or serial number specified by GOTO statement is not found in the blocks containing M98, M99, G65 and G66.	
079	A communicated program is not identical with the original one (program comparison).	
084	Program edit is impossible as a result of improperly instructed origin, end point or travel end point in expanded edit function.	
085	The number of bits or baud rate of input data is not correct during reading in with RS232 and DNC connectors.	
086	Transmission or I/O device is abnormal during input and output with RS232 connector.	
087	A data with more than 10 characters is input after sending DC3 stop code when RS232 and DNC connectors are used for reading in.	
090	The one-revolution signal from pulse coder (basic origin signal for linear graduation) is not input or reference point is not correctly returned to during the return to reference point in grid mode.	
091	The one-revolution signal from pulse coder (basic origin signal for linear graduation) is not in step with basic counter and reference point is not correctly returned to during the return to reference point in grid mode.	
092	The axes instructed by G27 cannot return to reference point.	
094	The restart of program cannot instruct type P (because coordinate system setting, clearing and other operations are performed after program interrupt).	
095	The restart of program cannot instruct type P (because external workpiece origin offset is changed after program interrupt).	
096	The restart of program cannot instruct type P (because workpiece origin offset is changed after program interrupt).	
097	The restart of program cannot instruct type P (because automatic operation has been never performed after power on or emergency stop and termination of storage limit alarm (emergency stop)).	

098	Reference point is not returned to after switching on or emergency stop and termination of storage type limit alarm (instant stop) and G28 is found in program search using program restart instruction.	
099	Move instruction in MDI mode is executed after the end of the search using program restart.	
100	The RESET button is pressed for reset by switching the parameter writing switch from ON to OFF.	
101	Power is disconnected when storing and editing a part program or rewriting the memory. In the event of alarm, the memory is completely cleared by concurrently pressing the DELETE and RESET buttons to switch on the power.	
102	Power is disconnected when writing in data of tool life management.	
110	The absolute value of data indicated by fixed point goes beyond the allowable range.	
111	The data displayed by floating point goes beyond the upper limit.	
112	The divisor is 0.	
113	A function that cannot be used for macro A is used.	
114	A format other than FOMAT is incorrect.	
115	A value that cannot be defined by variable is instructed.	
116	The left of the assignment statement is a prohibited variable.	
118	The number of layers of brackets goes beyond the upper limit (5).	
119	The independent of SQRT or BCD is negative, or a numeral other than 0 to 9 serves as a digit.	
122	The number of calling layers of macro goes beyond the permissible range (1-4).	
123	Macro control instruction is used in DNC mode.	
124	DO－END do not correspond one to one.	
125	The format of (FORMUCA) is incorrect.	
126	In DON, the value of N does not fall within 1≤N≤3.	
127	NC instructions are mixed with macro instructions.	
128	In GOTON, the value of N does not fall within 0≤N≤9999.	
129	A prohibited address is used in (independent variable assignment).	
130	The data of a big address is incorrect during the input of external data.	
131	5 or more alarm numbers appear in the external alarm information area.	
132	There is no corresponding alarm number in the clearance of external alarm information.	
133	The data of a minor address is external alarm information and external operating information.	
134	The rotary plane of coordinate system or the plane of arc or tool radius compensation C is incorrect.	
140	Tool group number goes beyond the upper limit (one of 16, 32, 64 and128).	

141	The tool group number instructed in machining program is not set.	
142	The number of tools in a group goes beyond the upper limit of storable number.	
143	T code is not stored in the block for setting tool group.	
144	H99 or D99 is specified when the tool in group is not used.	
145	The T code following M06 in program execution does not match the relevant T code in tool group in service.	
146	There is not P and L instructions at the beginning of the program for setting tool group.	
147	The set number of tool groups goes beyond the upper limit.	
148	The values of parameter No.333, 334 and 335 are beyond the setting range.	
160	An executing program is edited or a program to be edited is not selected. Sequence number is searched before edit.	
170	The programs of program number 8000 to 8999 and 9000 to 9899 are edited. However, these programs are prohibited from edit, resulting in the alarm (see parameters No.318 for PRG9 and 319 for PRG8)	
180	A value below the decimal point is specified when a decimal point is used in B-axis instruction and a value beyond the integral multiple of the minimum indexing angle of the indexing workbench is instructed.	Indexing job Indexing function.
181	One of the axes X, Y and Z is instructed at the same time with axis B.	
190	Axis is specified incorrectly in the control of surface constant speed.	
210	The moving part of the machine contacts the travel limit switch in + direction of axis X.	
211	The movable part of the machine contacts the travel limit switch in - direction of axis X.	
212	Tool enters the exclusion area of storage type travel limit 1 when axis X moves in the positive direction.	
213	Tool enters the exclusion area of storage type travel limit 1 when axis X moves in the negative direction.	
214	Tool enters the exclusion area of storage type travel limit 2 when axis X moves in the positive direction.	
215	Tool enters the exclusion area of storage type travel limit 2 when axis X moves in the negative direction.	
220	The movable part of the machine contacts the travel limit switch on the positive side of axis Y.	
221	The movable part of the machine contacts the travel limit switch on the negative side of axis Y.	
222	Tool enters the exclusion area of storage type travel limit 1 when axis Y moves in the positive direction.	
223	Tool enters the exclusion area of storage type travel limit 1 when axis Y moves in the negative direction.	
224	Tool enters the exclusion area of storage type travel limit 2 when axis Y moves in the positive direction.	
225	Tool enters the exclusion area of storage type travel limit 2 when axis Y moves in the negative direction.	
230	The movable part of the machine contacts the travel limit switch on the positive side of axis Z.	

231	The movable part of the machine contacts the travel limit switch on the negative side of axis Z.	
232	Tool enters the exclusion area of storage type travel limit 1 when axis Z moves in the positive direction.	
233	Tool enters the exclusion area of storage type travel limit 1 when axis Z moves in the negative direction.	
234	Tool enters the exclusion area of storage type travel limit 2 when axis Z moves in the positive direction.	
235	Tool enters the exclusion area of storage type travel limit 2 when axis Z moves in the negative direction.	
240	The movable part of the machine contacts the travel limit switch on the positive side of the 4th axis.	
241	The movable part of the machine contacts the travel limit switch on the negative side of the 4th axis.	
242	Tool enters the exclusion area of storage type travel limit 1 when the 4th axis moves in the positive direction.	
243	Tool enters the exclusion area of storage type travel limit 1 when the 4th axis moves in the negative direction.	
250	The movable part of the machine contacts the travel limit switch on the positive side of the 5th axis.	
251	The movable part of the machine contacts the travel limit switch on the negative side of the 5th axis.	
252	Tool enters the exclusion area of storage type travel limit 1 when the 5th axis moves in the positive direction.	
253	Tool enters the exclusion area of storage type travel limit 1 when the 5th axis moves in the negative direction.	
400	Axes X, Y and Z are overloaded.	
401	The speed ready signal (VRDY) of axes X, Y and Z are disconnected.	
402	Additional axis is overloaded.	
403	The speed ready signal (VRDY) of additional axis is disconnected.	
404	Though the position ready signal (PRDY) is switched off, the speed ready signal (VRDY) is not OFF. When the power is switched on again, the ready signal (PRDY) is not ON, so the speed ready signal (VRDY) will be ON.	
405	The malfunction in NC or servo system may prevent returning to reference point.	
407	The position ready signal (VRDY) of the 5th axis is switched off.	
410	The position offset during the stop of axis X exceeds the setting.	
411	The position offset during the travel of axis X exceeds the setting.	
412	Axis X drifts excessively (above 500VELO).	
413	The position offset of axis X exceeds ±32767 or the speed instruction value of DA converter is beyond +8191 to -8192. The alarm is often caused by wrong setting.	
414	The detecting device of the rotary transformer and inductosyn of axis X malfunctions.	
415	A speed above that in unit/sec detected by 511875 is instructed in axis X. The alarm is the result of incorrect CMR setting.	
416	The detecting device for the pulse coder of axis X is out of order (disconnection alarm).	

417	The servo position LSI of axis X is incorrect.	
420	Position offset exceeds the setting when axis Y stops.	
421	Position offset exceeds the setting when axis Y travels.	
422	Axis Y drifts excessively (over 500VELO).	
423	The position offset of axis Y exceeds ±32767 or the speed instruction value of DA converter goes beyond the range of +8191 to -8192. The alarm is usually a result of incorrect settings.	
424	The detecting device of rotary transformer and inductosyn of axis Y malfunctions.	
425	A speed above that in unit/sec detected by 511875 is instructed in axis Y. The alarm is the result of incorrect CMR setting.	
426	The position detector for the pulse coder of axis Y is out of order.	
427	The servo position LSI of axis Y is incorrect.	
430	Position offset exceeds the setting when axis Z stops.	
431	Position offset exceeds the setting when axis Z travels.	
432	Axis Z drifts excessively (more than 500VELO).	
433	The position offset of axis Z exceeds ±32767 or the speed instruction value of DA converter goes beyond the range of +8191 to -8192. The alarm is usually a result of incorrect settings.	
434	The detecting device of the rotary transformer and inductosyn of axis Z malfunctions.	
435	A speed above that in unit/sec detected by 511875 is instructed in axis Z. The alarm is the result of incorrect CMR setting.	
436	The detecting device for the pulse coder of axis Z is out of order (disconnection alarm).	
437	The servo position LSI of axis A is incorrect.	
440	Position offset exceeds the setting when the 4^{th} axis stops.	
441	Position offset exceeds the setting when the 4^{th} axis travels.	
442	The 4^{th} axis drifts excessively (more than 500VELO).	
443	The position offset of the 4^{th} axis exceeds ±32767 or the speed instruction value of DA converter goes beyond the range of +8191 to -8192. The alarm is usually a result of incorrect settings.	
444	The detecting device of the rotary transformer and inductosyn of the 4^{th} axis malfunctions.	
445	A speed above that in unit/sec detected by 511875 is instructed in the 4^{th} axis. The alarm is the result of incorrect CMR setting.	
446	The detecting device for the pulse coder of the 4^{th} axis is out of order (disconnection alarm).	
447	The servo position LSI of the 4^{th} axis is incorrect.	

450	Position offset exceeds the setting when the 5th axis stops.	
451	Position offset exceeds the setting when the 5th axis travels.	
452	The 5th axis drifts excessively (more than 500VELO).	
453	The position offset of the 5th axis Y exceeds ±32767 or the speed instruction value of DA converter goes beyond the range of +8191 to -8192. The alarm is usually a result of incorrect settings.	
454	The detecting device of the rotary transformer and inductosyn of the 5th axis malfunctions.	
455	A speed above that in unit/sec detected by 511875 is instructed in the 5th axis. The alarm is the result of incorrect CMR setting.	
456	The position detecting device for the pulse coder of the 5th axis is out of order (disconnection alarm).	
457	The servo position LSI of the 5th axis is defective.	
600	Data transfer error of the connection unit.	
601	Ready signal is cut off.	
602	PC program is not loaded (only PC-Model A)	
603	The communication between the NC and PC is improper or interrupted.	
604	The MPU of PC-B cannot be held or the PCGREADY or PC-GR cannot be switched ON.	
605	System alarm is given in the MPU or PC-B or G (monitor alarm).	
606	RAM/ROM parity error occurs in the MPU or PC-B or G.	
607	The data transfer error of MDI/LCD.	
700	The main PCB is overheated.	
701	The additional PCB is overheated.	
702	The DC motors of axes X, Y and Z are overheated.	
703	The motor of the 4th axis is overheated.	
704	The motor of the 5th axis is overheated.	
900	Non-volatile memory circuit 1 malfunctions.	
901	Non-volatile memory circuit 2 malfunctions.	
902	Non-volatile memory circuit 3 malfunctions.	
903	Non-volatile memory circuit 4malfunctions.	
904	Non-volatile memory circuit 5 malfunctions.	

905	Non-volatile memory circuit 6 malfunctions.	
906	Non-volatile memory circuit 7 malfunctions.	
907	Non-volatile memory circuit 8 malfunctions.	
908	Non-volatile memory circuit 9 malfunctions.	
909	Non-volatile memory circuit 10 malfunctions.	
910	RAM parity error (lower byte)	
911	RAM parity error (higher byte)	
912	Non-volatile memory circuit 11 malfunctions.	NO alarm
920	System alarm (fault of monitoring timer).	
930	CPU malfunctions (Class 0, 3 and 4 interrupts).	NO alarm
940	Offset memory alarm (excessive offset is set): Set correct offset in the specified offset number.	
950	Clock malfunctions (the clock of main PCB malfunctions).	
960	The temporary storage area for system control commands becomes inadequate (overflow).	
961	CPU alarm (execution of INT instruction)	
996	Necessary option of additional RAM is added, but RAM is not really installed.	
997	ROM parity error (PC ROM).	
998	ROM parity error (primary ROM).	
999	ROM matching error (height mismatching)	

Appendix 7 List of the States during Switching On, Reset and Clearance

O. A state remains unchanged or a movement is kept on.

X. A state is cancelled or a movement is interrupted.

Items		When switching on	Clearing state	Reset state
Setting data	Offset	○	○	○
	Setting data	○	○	○
	Parameter	○	○	○
Data	Programs in memory	○	○	○
	Information in memory	×	×	In MDI mode ○ Other than in MDI mode ×
	Indication of sequence numbers	×	○(Note 1)	○(Note 2)
	One-off G codes	×	×	×
	Modal G codes	Initial value G20/G21 remains constant as before power failure	Initial values G20/G21 and G22/G23 remain unchanged.	All remain unchanged.
	F	Zero	Zero	○
	S·T·M·B	×	○	○
	L	×	×	In MDI mode ○ Other than in MDI mode ×
Coordinate system	Workpiece coordinates	Zero	○	○
Executing motions	Travel	×	×	×
	Dwell	×	×	×
	M·S·T·B code is sent.	×	×	×
	Tool length compensation	×	Depending on “RS43”	MDI mode ○; a mode other than MDI depends on “RS43”

(Note 1) Program number is displayed from the beginning of the program.

Items		When switching on	Clearing state	Reset state
Executing motions	Tool radius compensation	×	×	In MDI mode ○ Other than in MDI mode ×
	The storage of the called subprogram numbers	×	× (Note 2)	In MDI mode ○ Other than in MDI mode × (Note 2)
Indicators and output signals	ALM	The indicator goes out in the absence of alarm.	See the left.	See the left.
	NOT READY	×	× (The indicator is lit in case of emergency stop.)	×(The indicator is lit in case of emergency stop.)
	LSK	The indicator is lit.	The indicator is lit.	MDI mode: ○ A mode other than MDI: the he indicator goes out.
	BUF	The indicator goes out.	The indicator goes out.	MDI mode: ○ A mode other than MDI: the he indicator goes out.
	Return to reference point	×	○(emergency stop×)	○(emergency stop ×)
	S·T·B codes	×	○	○
	M codes	×	×	×
	M·S·T·B strobe signal	×	×	×
	Spindle rotation signal (S12-digit /S analog signal)	○	○	○
	NC ready signal (MA, MB)	ON	○	○
	Servo ready signal	ON (in the absence of servo alarm)	See the left.	See the left.
	Signal indicator in CYCLE START	×	×	×
	Signal indicator in feed hold	×	×	×

Note 2: When the NC is reset in the execution of a subprogram, the next block of the called subprogram during the return of the control to the main program cannot be executed in midway due to the subprogram. Therefore control returns to the start of the program.

Appendix 8 Memory Type Pitch Error Compensation

A8.1 Feature

Pitch error compensation is applicable for the minimum instruction units for all axes. The function is valid after returning to reference point.

A8.2 Specifications

The tool position after returning to reference point is called compensation origin. The compensations of the concerned axes are set in parameters.

i) Compensable axes: axes X, Y and Z as well as the 4th and 5th axes.

ii) Number of compensation points:

Linear axis – 128 points

Rotary axis – 61 points

iii) Range of compensation

0 to ± 7X compensation scaling rate/compensation point (minimum instruction unit)

Compensation scaling rate X1, X2, X4 and X8 (common for all axes)

iv) Compensation interval

Travel unit	Minimum set interval	Maximum set interval	unit
Metric system	8000	20000000	0. 001mm
inch system	4000	20000000	0.0001inch

(Maximum compensation range = set interval ×128)

Actual compensation interval shall be set depending on the optimal value between the maximum compensation distance and mechanical travel in the ranges as listed in the above table. When the 4th or 5th axis is used as an axis of rotation, compensation interval is set within 360000.

In this case, a value less than 110,000deg/min (31.2rpm) shall be used for the feedrate of an axis of rotation.

When the set interval is less than the above minimal set interval, compensation is impossible for a linear axis.

Now it is necessary to reduce the rapid feedrate.

A8.3 Parameter setting

Parameters concerning pitch error shall be set in the following parameter numbers in MDI mode or emergency stop mode.

(1) Pitch error compensation scaling rate

	0	2	4

PML2	PML1	※	※	※	※	※	※

The scaling rate is multiplied by the set compensation for output.

PML2	PML1	Scaling rate
0	0	X 1
0	10	X 2
1	0	X 4
1	1	X 8

(Common for all axes)

(2) Pitch error origins

	0	3	9

P E C Z R X

	0	4	0	P E C Z R Y
	0	4	1	P E C Z R Z
	0	4	2	P E C Z R 4
	4	1	6	P E C Z R 5

Pitch error origins PECZRX, Y, Z, 4 and 5 specify the origin values in the pitch error compensations list.

The setting ranges of all axes are any value between 0 and 127 depending on machine type.

(3) The setting of compensation interval

	1	6	3	PECINTX
	1	6	4	PECINTY
	1	6	5	PECINTZ
	1	6	6	P E C I N T 4
	4	3	6	P E C I N T 6

Pitch error compensation intervals PECINT X, Y, Z, 4 and 5 specify pitch error compensation intervals.

Except setting the positive values of 8000 or above (metric system) and 4000 or above (Inch) (for the value that the axis of rotation is 3), the setting of 0 is not compensated.

(4) Compensation setting

The pitch error compensation for all axes is set in the following parameters:

Axis name	Parameter number
Axis X	1000 to 1127
Axis Y	2000 to 2127
Axis Z	3000 to 3127
The 4th axis	4000 to 4127
The 5th axis	5000 to 5127

Compensation cannot be set for the parameter numbers other than listed in the above table. The set compensation rage is 0±7. The setting beyond the range is invalid.

(Example)

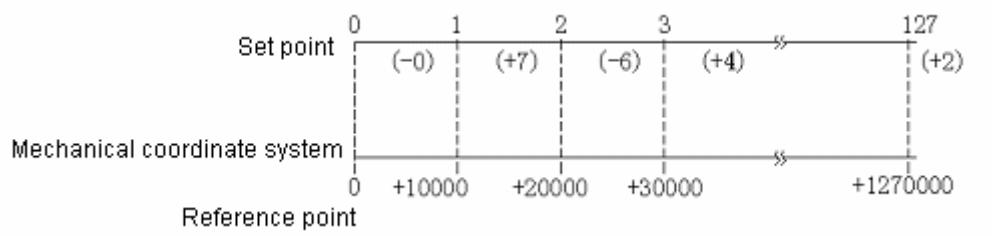
1	1	2	0

——							7
7	6	5	4	3	2	1	0

In the above example, a compensation of ——7 is set at the set point No.120 of axis X. The value of the set point is added or reduced by 1 using the cursor [↑] or [↓]. The compensation values at adjacent points are displayed.

A8. 4 Examples of all parameter settings

(1) Example 1: Pitch error origin =○, compensation interval =10000



The beginning of the compensation list corresponds to the reference point while compensation point 1 corresponds to the point moving by 10000 in the positive direction from the reference point. After that, every 1000 corresponds to a compensation point. As a result, the compensation point128 is set at 1270000. The compensation for the travel from 0 to 10000 is set at compensation point 1 while that from 1000 to 2000 is set at compensation point 2 and that from (n −1)× (compensation interval) corresponds to n×(compensation interval) is set at compensation point n.

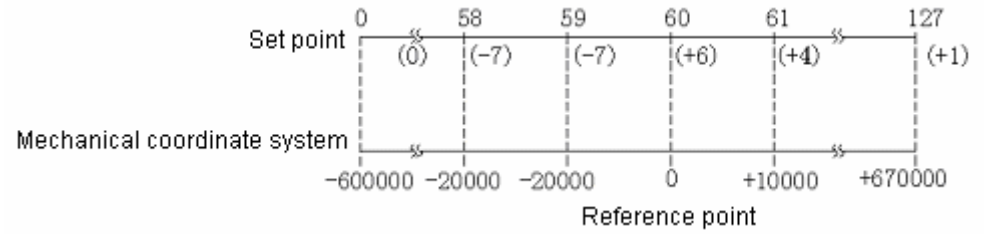
In the above example

Interval	Compensation data
0 to 10000	−7
10000 to 20000	+ 6
20000 to 30000	−4

When the real machine move from the reference point to the position at +30000, the total pitch error compensation is as follows:

(+7) + (-6) + (+4) = 5

(2) Example 2: Pitch error origin =60, compensation interval =10000



The set point 60 corresponds to the reference point. The set point 61 is located in the position of +10000. After that, every 10000 corresponds to a compensation point. The set point 59 is located in the position of -10000. After that, every -10000 corresponds to a compensation point and compensation point 0 is located at -600000. That is, the compensation for the travel from (n-61) × (compensation interval) to (n-60)×(compensation interval) is set at compensation point n.

In the above example, in the interval

-30000 to -20000: A compensation of +7 is set.

-20000 to 10000: A compensation of +7 is set.

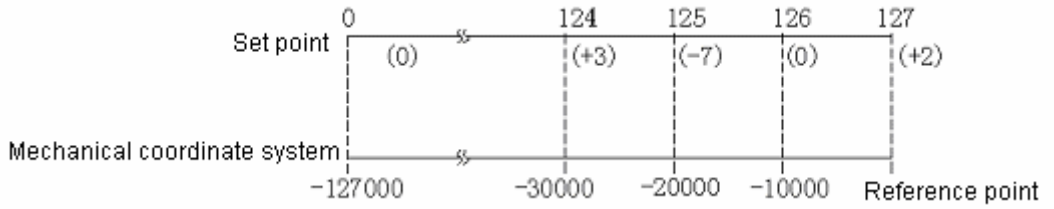
-10000 to 0: A compensation of -6 is set.

0 to 10000: A compensation of -4 is set.

In the above example, the total compensation for pitch error when the machine moves from -30000 to +10000 is:

(-7)+(-7)+(+6)+(+4)=(-4)

(3) Example 3: Pitch error origin =127, compensation interval =10000



The end of the compensation list corresponds to the reference point. The set point 126 is located in the position of -10000. After that, every -10000 corresponds to a compensation point. The compensation for the travel from -10000 to 0 is set at compensation point 127 while that from -20000to -10000 is set at compensation point 126. The compensation for the travel from (n-128) × (compensation interval) to (n-127)×(compensation interval) is set at compensation point n.

In the above example, compensation data is as follows:

in the interval -40000 to -30000: -3

-30000 to -20000: +7

-20000 to 10000: 0

-10000 to 0: -2

The total compensation for pitch error when the real machine moves from -40000 to the reference point is:

(+3)+(-7)+(0)+(2)=(-2)

A8. 5 Compensation setting method

The relationship between the compensation as shown in the above item and the reference point as well as compensation origin

- The traveling direction of the machine
- Compensation interval

They are not directly related. The compensation at the compensation point n (n=0, 1, 2……127) is determined by the mechanical error (the remaining travel for a move instruction) in the interval { n-(compensation origin +1) } ×compensation interval to (n — compensation origin)×(compensation interval).

(1) Method for inputting compensation

The input of compensation is possible by the identical means of the common parameter input.

(a) Cancellation of compensation

For an axis eliminated of compensation, all the compensations of the axis are "○" after a compensation of -9999 is input in any parameter number.

(b) Output of compensation

All compensations can be output in the sequence identical with that of common parameter output. However, -9998 rather than -9999 shall be typed now. This type of output is impossible for some specific axes.

(2) Precautions for setting

(a) When the setting of compensation interval for the setting (parameters No.163 to 166 and 436) is a positive value, the value will be used for compensation.

When it is a negative value, its absolute value will be used for compensation.

When it is 0, the axis will not be compensated.

(The indication is positive even a negative compensation interval is input)

(b) Pitch error compensation is valid at the end of return to reference point. If compensation is not made after parameter setting at the end of return to reference point, parameter setting shall be made before returning to reference point after switching on. If pitch error

compensation is changed after returning to reference point, the reference point shall be returned to again.

(c) Pitch error compensation (parameters No.1000 to 5127)

The following restrictions are made for pitch error compensation:

The value of (Valid compensation of pitch error)×(pitch error compensation scaling rate)×CMR must fall within ±127.
If the setting of the value goes exceeds ±127, correct compensation cannot be made. If it is necessary to set a value beyond ±127, divide the compensation and then make compensation at adjacent points.
Note: CMR: Command multiplying rate

Refer to parameters No.27, 28, 29 and 30.

A8.6 Pitch error compensation for an axis of rotation

For pitch error compensation for the 4th axis as an axis of rotation, its parameters are set as follows:

Parameter No.	Descriptions	Parameter setting
42	Compensation zero	0
166	Compensation interval	6000

That is, pitch compensation error origin =0 and compensation interval=6000 shall be used. Now a circumference is divided into 60 parts and compensation is made for every 6 degrees.

Compensation is set as the following 61 points.

Parameter No.	Parameter setting
4000	-6 to 0 deg compensation
4001	0 to 6 deg compensation
4002	6 to 12 deg compensation
⋮	⋮
4059	348 to 354 deg compensation
4060	354 to 360 deg compensation

In special cases, parameters 4000 and 4060 may be set to the same value. The pitch error compensation for an axis of rotation uses rapid feedrate 110000deg/min (31.2rpm) or the following speeds.

The sign of the pitch error compensation is based on its moving direction.

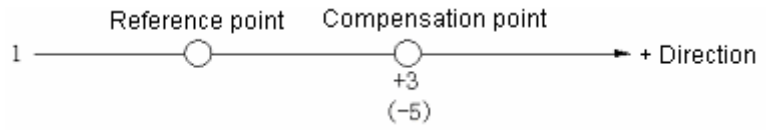
Namely for positive compensation, the amount of travel is only increased by one quantity of compensation.

For negative compensation, the amount of travel is only reduced by one quantity of compensation. When mechanical travel has negative and positive errors opposite to the move instruction, negative and positive compensations are set.

When the error (excessive travel) is positive (+), the compensation will be negative (-).

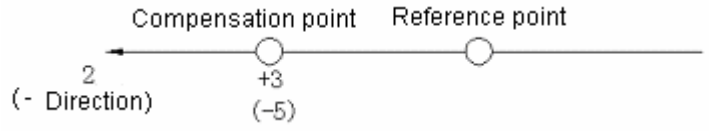
When the error (excessive travel) is negative (-), the compensation will be positive (+).

(Example of setting)



For travel in positive direction, +3 (-5) shall be compensated by the arrival of the compensation point.

For travel in negative direction, -3 (+5) shall be compensated by the arrival of the compensation point.



For travel in positive direction, +3 (-5) shall be compensated by the arrival of the compensation point.

For travel in negative direction, -3 (+5) shall be compensated by the arrival of the compensation point.

(Note) Thus it can be seen that the sign of compensation is only related to the traveling direction during compensation instead of the position of origin.

Appendix 9 Operations List

Item	Function	Program protection OFF	Parameter writing ON/OF	Mode	Function key	Operating procedures
Clear	To clear memory		○	Power ON	—	Concurrently press the O and DELETE buttons to switch on the power.
	To clear parameters		○	Power ON	—	Concurrently press the CANCEL and DELETE buttons.
	To clear programs			Power ON	—	Concurrently press the RESET and DELETE buttons.
Communication input	To input parameters		○	Emergency stop ON	PARAMETER	P → - 9999 - INPUT
	To store a program	○		EDIT mode		(O → program number →INPUT
	To add programs	○		EDIT mode	—	O→ CANCEL →INPUT
	To store all programs			EDIT mode	—	O → - 9999 - INPUT
	Parameters for pitch error compensation		○	Emergency stop ON	PARAMETER	P → - 9999 - INPUT
MDI input	To input parameters		○	MDI mode	PARAMETER	N →parameter number→ INPUT → P →data→INPUT parameter writing switch OFF→ RESET (Note 2)
	To input an offset	○		Any mode (Except EDIT)	OFFSET	N →offset No.→ INPUT → P →offset data→INPUT
	To input setting data	○		MDI mode	SETTING	Move the cursor to the setting number to changed→ P → data→INPUT
Communication output	To output parameters			EDIT mode	PARAMETER	P → - 9999 - OUTPUT
	To output an offset			EDIT mode	OFFSET	P → - 9999 - OUTPUT
	To output the parameters for pitch error compensation			EDIT mode	PARAMETER	P → - 9999 - OUTPUT

	To output all programs			EDIT mode	—	O → - 9999 - OUTPUT
	Output a program.			EDIT mode	—	O → program No. - OUTPUT
Search	Program number search (stored in memory)			EDIT mode	PARAMETER	① O →program No→ ↓ (cursor ② O → CANCEL → ↓(cursor)
	Sequence number search (stored in memory)			AUTO mode	PARAMETER	Program number search→ N →sequence number→↓(cursor)
	Word search (stored in memory)			EDIT mode	PARAMETER	Input the address and data to be searched→ ↓(cursor)
	Address search (stored in memory)			EDIT mode	PARAMETER	Input the address to be searched→↓(cursor)
Edit	To delete all programs	○		EDIT mode	PARAMETER	O → - 9999 - DELETE
	To delete a programs	○		EDIT mode	PARAMETER	O →program number - DELETE
	To delete several programs	○		EDIT mode	PARAMETER	N →sequence number - DELETE
	To delete a single program	○		EDIT mode	PARAMETER	Search to the beginning of the block to be deleted→ EOB →DELETE
	To delete a word	○		EDIT mode	PARAMETER	Search to the word to be deleted→ DELETE
	To change a word	○		EDIT mode	PARAMETER	Search to the word to be changed→ address → data— ALTER
	To insert a word	○		EDIT mode	PARAMETER	Search to the word preceding the one to be inserted→ address →data→ INSERT
	Memory sorting	○		EDIT mode	PARAMETER	CANCEL → SHIFT

Appendix 10 Lock of Program Key

A10.1 General

Program numbers 9000 to 9899 are locked with keys. In key lock mode, the programs of program numbers 9000 to 9899 cannot be displayed, edited and output. The function may be used to protect the special programs developed with manufacturer's user macros and prevents them from accidental deletion.

A10. 2 Program number

It is possible to lock the programs of program numbers 9000 to 9899 with keys. Other programs cannot be locked with keys. Once they are locked with keys, all the programs of program numbers 9000 to 9899 are automatically locked. Therefore, the programs that do not need key lock shall use the numbers other than 9000 to 9899.

A10. 3 The state after key lock

In key lock mode (see Section 4 below), the programs of program numbers 9000 to 9899 are as follows:

(1) Their information is not displayed even in execution.

(2) Program number search (alarm No.071) is impossible in EDIT mode (AUTO mode). Therefore they cannot be edited.

(3) The memory cannot be cleaned up.

(4) The numbers are not included in the display of all program numbers.

(5) Program output is impossible (not output even when all programs are output).

(6) Program deletion is impossible (not deleted even when all programs are deleted).

(7) Program storage is impossible (alarm No.170).

A10. 4 Key locking and unlocking procedures

(1) Preset a secret number (1 to 99999999) in parameter No.168. Do not to forget the number since the contents of the parameter is not displayed. A program cannot be unlocked if it is set to 0.

(Note 1) The setting of the parameter is active in unlock state.

(Note 2) The parameter is not deleted even in complete clearing state.

(Note 3) The parameter becomes 0 after complete clearance of memory, i.e. the key is disabled.

(2) To disable the key, set the same information in parameter No.408 and 168.

The key is disabled only when it has the same value with parameter No.168.

(Note 1) The settings of the parameter are not displayed.

(Note 2) The parameter cannot be stored in non-volatile memory.

(3) Method of key lock after unlock

(a) Set a different value in parameter No.408 than in parameter No.168.

(b) Switch off the NC power and then switch on it again.

Parameter No.

	1	6	8	Secret No.

Store the secret number to be locked.

Setting range: 1 to 99999999

	4	0	8	Lock/unlock

The key lock is disabled by entering the same value as No.168.

The key lock is active when a different value is entered.

(Note 1) When the value of parameter No.168 is set to 0, the key lock is disabled and is inactive even the power lock is switched OFF. Note that parameter No.168 must be set to 0 when key lock is not necessary.

A10. 5 Cautions

(1) Proceed as follows if the set secret code is forgotten:

(a) Completely clear the memory (disable key lock).

(b) Input all parameters (except parameter No.168)

(c) Save secret programs in the memory.

(d) Set a secret code in parameter No.168 (with key lock).

(2) The PROGRAM page is displayed as follows if a program with key lock is switched to EDIT mode. The settings of the program are hidden. The display or edit of other programs shall be performed by reset (now continuous operation is impossible due to reset). The initial page will be returned to once the system is reset. To continue to execute a program, switch the mode to AUTO or DNC mode. Machining is performed using CYCLE START.

(3) Once programs ○90000 to ○9899 are stored and edited, please apply key lock for the programs other than ○9000 to ○9899. When programs ○9000 to ○9899 are displayed, the above page is displayed in EDI mode and programs are displayed by reset.

Appendix 11 The Interrupt Function of User Macro

A11.1 General

During the execution of a program, it is possible to call another program by inputting interrupt signal on the machine side. This function is called interrupt function of user macro.

The interrupt instruction in a program is as follows:

M96, P××××；user macro interrupt ON

M97, P××××；user macro interrupt OFF

By using this function it is possible to call other programs on any executing block of program and start program operations in ever-changing conditions.

(Example of application)

(1) Start tool abnormality detection with external signal.

(2) Stop the currently performing machining and insert other machining in the continuous machining.

(3) The current machining information is read regularly.

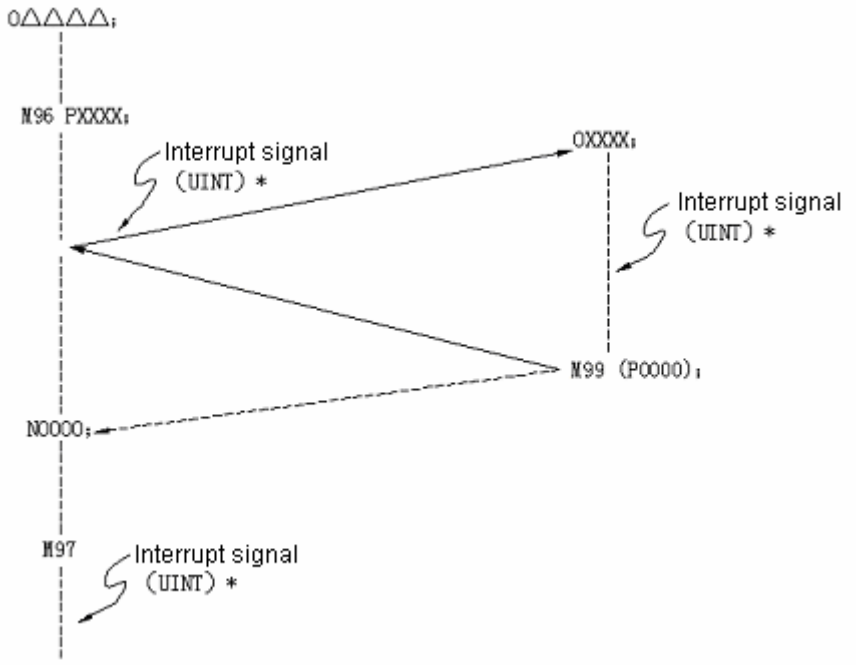


Fig.1: Schematic diagram of user macro interrupt

It is also applicable for the applications of adaptive control.

After instructing M96P×××× in program, the currently executing program is interrupted once interrupt signal (UINT) is input.

Programs instructed by P××××

The interrupt signal during the execution of program and after M97 is invalid (marked "*" in the figure).

A11.2 Method of instruction

A11.2.1 Significant conditions

User macro INTERRUPT is only effective during the execution of a program.

Namely the significant conditions are:

(1) MEMORY, TAPE or MDI mode is selected.

(2) STL (start indicator) is set to ON.

(3) User macro INTERRUPT is still not being executed. User macro INTERRUPT cannot be executed during JOG (JOG, STEP, HANDLE etc) operation.

A11.2. 2 Instruction types

User macro interrupt function disables or enables interrupt signal (UINT) using M96 and M97 in principle.

That is, when M96 is instructed, interrupt signal (UINT) may be used to start user macro INTERRUPT until M97 is instructed or NC is reset. Also, INTERRUPT (UINT) is invalid without the instruction of M96. User macro INTERRUPT cannot be started even by inputting interrupt signal (UINT) after M979 is instructed or reset.

(Format)

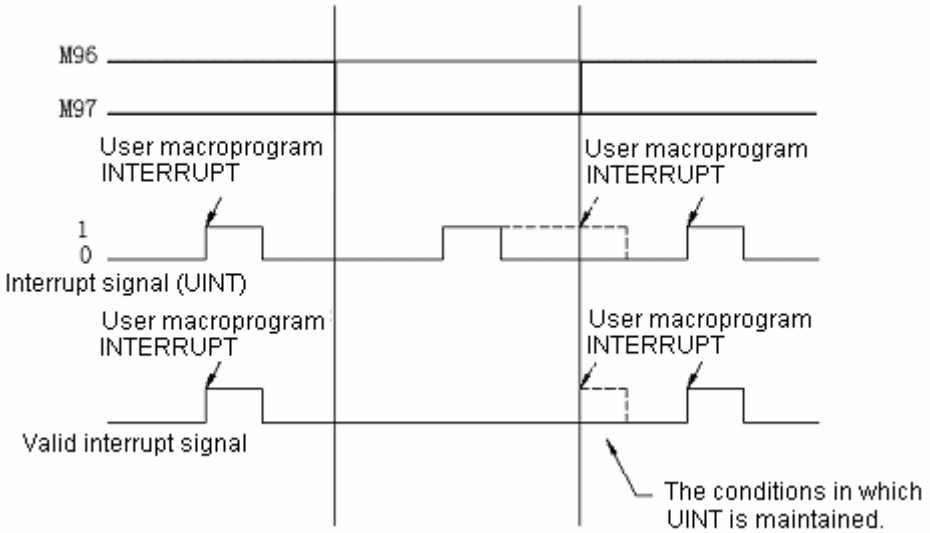
M96 P××××; User macro INTERRUPT ON

Instructing a program number to be interrupted

M97; User macro INTERRUPT OFF

Interrupt signal (UINT) is valid after the input of M96.

Once M97 is instructed, interrupt signal will be invalid. The signal input after instructing M97 is kept until the input of M96. Once M96 is instructed, user macro INTERRUPT can be started.



A11.3 Detailed descriptions

A11.3. 1 The ENABLE/MASK of user macro INTERRUPT

Even user macro INTERRUPT is not used, it is not necessary to change the program. For this purpose, parameter ENABLE/MASK (parameter No.025——MSUR) is provided for selecting user macro INTERRUPT.

If user macro INTERRUPT is set to MASK in the parameter, M96 and M97 will be output to the outside as common M codes. If it is set to ENABLE, they are processed inside the parameter without inputting to the outside.

A11.3. 2 Subprogram type interrupt and macro type interrupt

The modes of user macro interrupt include subprogram type interrupt and macro type interrupt. Therefore, the parameter No.025——MSUB is designed for an interrupt mode.

- Subprogram type interrupt

  Interrupt program is called by a subprogram.

  That is, the values of local variables remain unchanged before and after interrupt. In addition, the interrupt is not counted as the number of calls.

- Macro type interrupt

  Interrupt program is called as a macro.

  That is, the values of local variables change before and after interrupt.

  In addition, the interrupt is not counted as the number of calls. The number of calls of programs and macros executed in interrupt program are accumulated to their numbers respectively.

  Variables cannot be obtained from the executing programs even a user macro interrupt is a macro interrupt.

A11.3.3 M codes for control of user macro interrupt

In principle, M96 and M97 are used for the control of user macro interrupt. However, they may be used for other purposes upon manufacturer's requirements (M function and macro call). Therefore, whether these M codes are valid or not depends on the setting of parameter No.025——MPRM.

When the M codes for control of user macro interrupt are set as a parameter:

User macro is set to ON and the M codes are set in #053.

User macro is set to OFF and the M codes are set in #054.

When the parameter MPRM is 0, M96 and M97 become the M codes for control of user macro interrupt as well as #053 and #054.

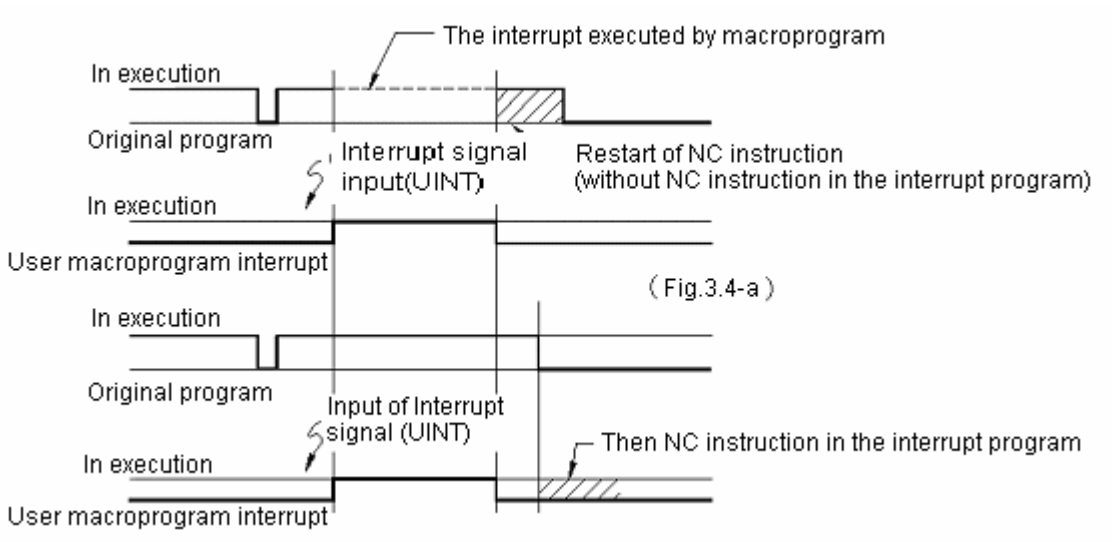
It is recommended not to use the M codes other than M96 and M97 for the control of macro interrupt in consideration of the interchangeability of program.

A11.3.4 User macro interrupt and NC instruction

User macro interrupt is of two types: interrupting the executing NC instruction and waiting for the end of the currently executing program. For this purpose, parameter No.314——MINT is designed for switching between interrupting in the midway/at the end of block.

If interrupting in the midway (type I) is selected in the parameter.

(1) Once interrupt signal (UINT) is input, the executing travel or suspension is interrupted and the interrupt program is executed.



(2) When there is an NC statement in the interrupt program, the interrupted program instruction disappears and the NC instruction in the interrupt program is executed. In case of return, the next block of the original program is executed.

(3) If there is no NC instruction in the interrupt program, the interrupted block is continuously executed when M99 is used to return to the original block.

If the parameter is selected, it is interrupted (type U) at the end of a block:

(1) Once interrupt signal (UINT) is input, the interrupt program instead of the instruction of the currently executing block is interrupted.

(2) When there is an NC instruction in the interrupt program, it is executed at the end of the executing program.

In any case, the control switches to the interrupt control program once interrupt signal is input.

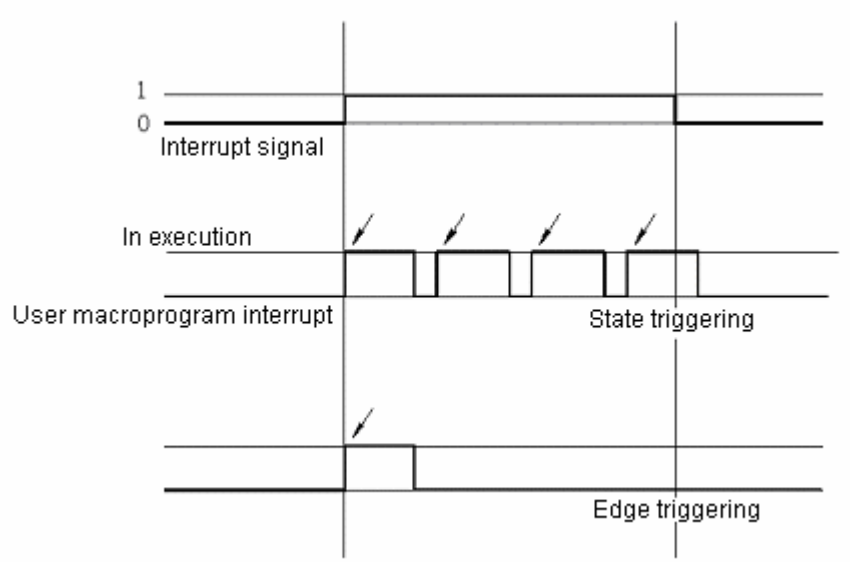
The relationship between the interrupt in the midway of program (Fig.3.4 – a) and the interrupt at the end of program (Fig.3.4 – b) are as indicated in the figure. Interrupt will be executed whichever interrupt signal is input.

A11.3. 5 Reception of user macro interrupt signal (UINT)

There are two modes of reception of user macro interrupt signal (UINT): state triggering and edge triggering. The so-called state triggering is that it is valid in the ON state of signal. Edge triggering is that the signal on the edge is active when the signal is switched from OFF to ON. The use of a mode shall be determined by parameter No.025——TSE.

If this parameter is set to state triggering mode, user macro interrupt occurs when the interrupt is active, i.e. the interrupt signal (UINT) is ON (1). Therefore, interrupt program can be repeatedly executed when the interrupt signal (UINT) is continuously ON.

In addition, when the parameter is set to edge triggering mode, interrupt program is completed in an instant (only the program instructed by macro, etc) because it is only valid during the rise of the interrupt signal (UINT). Therefore, it is only applicable for the occasions that is not suitable for state triggering mode and only one user macro interrupt is performed in the whole program (interrupt signal is kept ON in this case).



Except special purposes, the actual effects of the two modes are the same (there is not difference in the time from the input of signal to the interrupt of actual execution).

State triggering performs user macro interrupt in the ON state of signal. Edge triggering performs user macro interrupt in the rise of signal. Therefore, in the above example state triggering performs 4 interrupts while edge triggering only one.

A11.3.6 Return from user macro interrupt

The instruction for returning from user macro to the original program is M99. The sequence number in the returned program is also specified with address “P”. In this case, search starts from the relevant program switch and returns to the initially appeared program number (the same as “98”).

Other interrupts cannot be performed in the execution of user macro interrupt program. M99 may be used to clear the state. The conditions individually* instructed with M99 are executed before the end of the execution of the foregoing programs. Therefore user program interrupt is also valid for the last instruction of interrupt program. If it is not applicable, user program interrupt may be controlled with M96/M97.

*: The single block of M99, which only includes address

For O, N, P, L and M, the block is considered as the same block as the previous one of the program. Thus even a signal block does not stop. The program is:

G×× X××× ; M99 is actually identical with

G×× X××× ; M99.

(They vary in whether G×× executes or not before M99.)

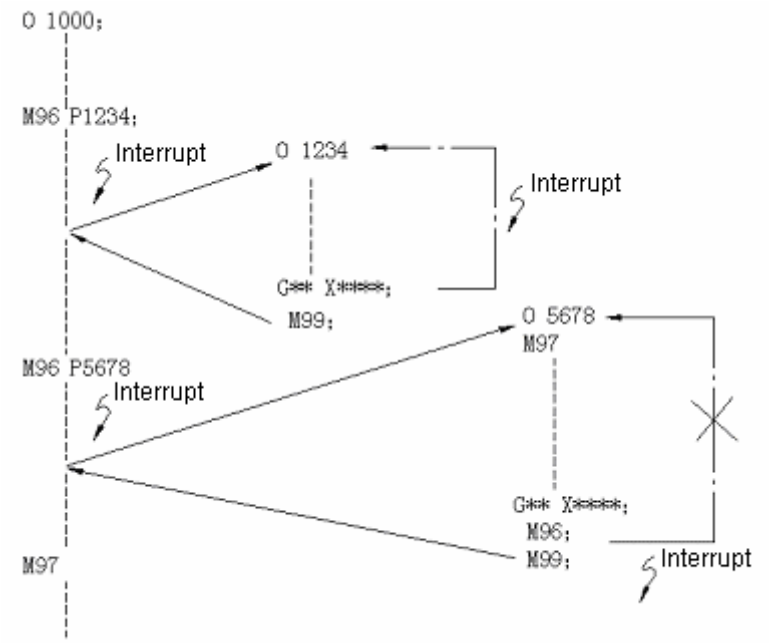


Fig. 3.6 Return from user macro

Overlapping is not allowed in the execution of user macro interrupt. That is, other interrupts are automatically shielded in case of interrupt. If M99 is executed, user macro interrupt will be active again. Here M99 is executed before the end of the previous program as a separate block. In the above example, the interrupt in the Gxx block of program 0 1234 is also valid. 0 1234 is executed after signal input.

In addition, 0 5678 is under the control of M96/M97. Hence the interrupt is valid only after returning to 0 1000.

A11.3.7 User macro interrupt and modal information

User macro is different from general program calls. It is started by interrupt signal (UINT) in program execution. As a rule, change of the modal information in an interrupt program has a negative impact on the original program. Therefore, modal information restores the state before interrupt when it returns to the original program with M99 even the modal information is changed in an interrupt program.

When M99 P×××× is used to return to the original program from an interrupt program, the modal information in a program is controllable. Hence the modal information changed in the reception of interrupt program. (On the contrary, when it is to receive the modal information in the original program, the return to the following travel is changeable depending on the modal data during interrupt.)

Therefore, in this case:

1) Modal information is given in the interrupt program.

2) Necessary modal information is instructed at return point. Application is taken into account like this.

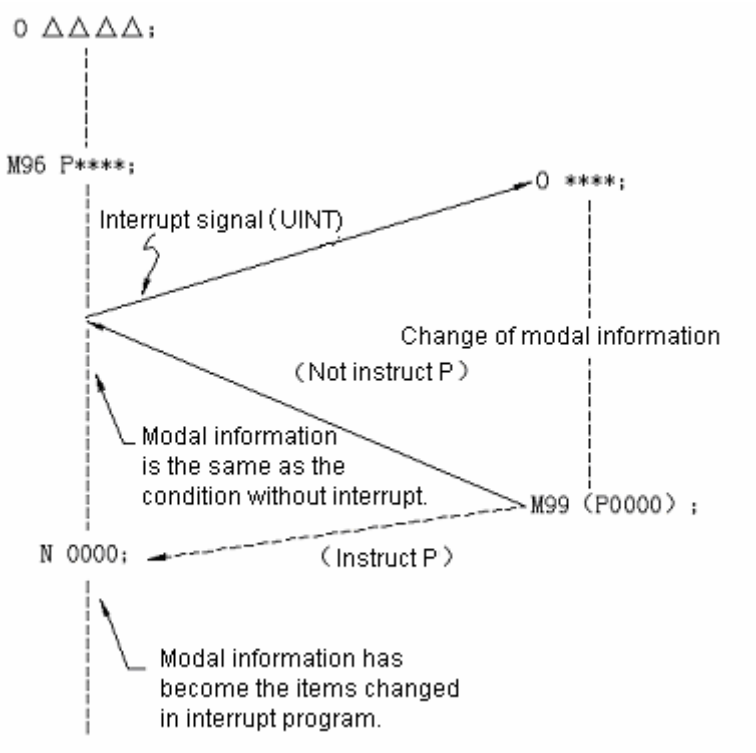


Fig. 3.7 User macro interrupt and modal information

Modal information shall be changed in interrupt program.

1) During the return of M99, the reception of modal information before interrupt is valid while the modal information changed in interrupt program is invalid.

2) During the return with M99 and P0000, the modal information changed in interrupt program is also valid after the return of modal information (the same as M98).

A11.4 Parameters

	0	2	5

7	6	5	4	3	2	1	0
MUSR		MSUB	MPRM				TSE

MUSR 1. Use macro interrupt function

0. Not use macro interrupt function

MSUB 1. Subprogram type user macro interrupt

0. Macro type user macro interrupt

MPRM 1. The M code for the control of user macro interrupt is set by parameter.

0. User macro interrupt is under the control of M96 and M97.

Note) User macro interrupt is a part of User macro B function. That is, user macro interrupt cannot be used without the selection of user macro B.

The settings of parameters No.053 and 054 are valid only when MPRM=1 (user macro B must be selected).

TSE 1. User macro interrupt is of state triggering type.

0. User macro interrupt is of edge triggering type.

Note) State triggering state is valid in signal input state.

Edge triggering is active in the rise of signal.

	0	5	3	MACINTON

MACINTON: The M code when user macro interrupt is active.

Settings 03 to 97

	0	5	4	MACINTOF

MACINTOF: The M code when user macro interrupt is inactive.

Settings 03 to 97

Note) Parameters No. 053 and 054 are valid only when MPRM (025Bit4)=1.

	3	1	4

7	6	5	4	3	2	1	0
	MINT						

MINT 1. The NC instruction that starts to execute interrupt program until the end of block (user macro interrupt type II)

0. The NC instruction that starts to execute interrupt program before the end of block (user macro interrupt type I)

Note) Once user macro interrupt signal (UINT) type I is input, the currently executing program is interrupted and the interrupt program is executed. The motion after the return varies with the availability of NC instruction in interrupt program.

i) With NC instruction

The remaining instructions (amount of travel and suspension time) disappear in the interrupted blocks.

ii) Without NC instruction

The remaining instruction in an interrupted block continues to execute. However, the sending auxiliary function can be correctly output in the both cases.

- Type II: The current block is not interrupted and the interrupt program executed even interrupt signal is input. If the interrupt program has an NC instruction, it starts to execute after the execution of the interrupted program.

A11.5 Diagnosis

	1	2	0

7	6	5	4	3	2	1	0
				UINT			

UINT 1. User macro interrupt ON.

0. User macro interrupt OFF.

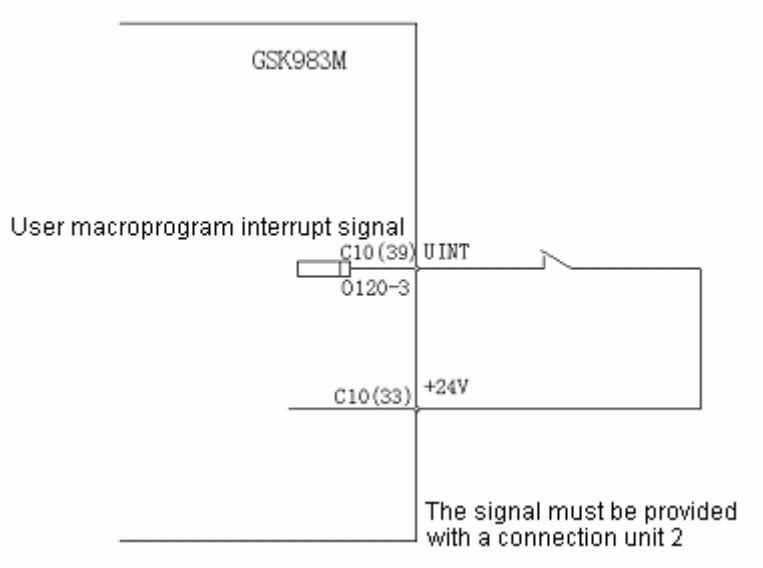
The signal is the external signal for user macro interrupt. Intended for the applications have requirements for high speed, the signal may be detected through not only the signal generated by strong power, but also external signal. Therefore, the manufacturer needs to connect the external signal to the position of the signal as shown below. The position is not applicable for other purposes.

	0	4	4

7	6	5	4	3	2	1	0
				UINT			

The diagnosis numbers to be observed are #044 and #120.

A11.6 Internal wiring diagram



A11.7 Example of application

A11.7.1 Procedures for starting tool fault detection using external signal

(Specifications)

Malfunction restoration is immediately executed even in cycle movement.

(Parameter setting)

TSE=0: Edge triggering mode

MUSR=1: Enable user macro interrupt function

MSUB=*: Subprogram type/macro type user macro interrupt

MPRM=*: Setting of the M code for the control of user macro interrupt
MINT=0: Interrupt program is executed before the end of block.
(Explanations)

User macro interrupt signal is ON during fault detection. It is kept on unless special operations are performed. Only one interrupt is performed if the edge triggering mode is selected. Whether the execution is interrupted or not is detected with diagnosis No.#120.

Parameters are set for the selection of subprogram type, macro type and control M code depending on programming.

A11.7.2 Inserting other job in continuous machining without interrupting the current program

(Specifications)

A short job is inserted in the program execution with longer machining time. General single block stops when the original program restarts. It is hard to execute the intervention in MDI mode.

(Parameter setting)

TSE=*: Selection of a triggering mode for user macro interrupt

MUSR=1: Enable user macro interrupt function

MSUB=0: Macro type user program interrupt

MPRM=*: Setting of the M code for the control of user macro interrupt

MINT=0: Interrupt program is executed after the end of block.

(Explanations)

Parameters are set for the triggering mode of user macro interrupt and the selection of M code as required.

For user macro interrupt, interrupt is prohibited in the execution of block and macro type interrupt is used to prevent the affection of the instruction segments in machining. The modal information, mechanical position, etc in case of interrupt in an interrupt program are restored during the return to the original program so that any program can by executed. To fix it, instruction M96 P×××× may be directly used. The program is called from the interrupt program with M98 P#100 if it is not fixed.

A11.7. 3 Reading machining information in fixed intervals

(Specifications)

To manage machining status, machining information is sent out on a regular basis. This exerts no impact on machining sequence.

(Parameter setting)

TSE=0: Edge triggering mode

MUSR=1: Enable user macro interrupt function

MSUB=0: Subprogram type user macro interrupt

MPRM=*: Setting of the M code for the control of user macro interrupt

MINT=0: Interrupt program is executed before the end of block

(Explanations)

Assuming that an interrupt program does not include any NC statement, the interrupt program is started only once in edge triggering mode on an regular basis depending on the ON/OFF of interrupt signal (because interrupt program is repeated when the signal is ON in state triggering mode). Since block may interrupt in the midway, the rise corresponding to the interrupt signal immediately interrupts.

The external output of machining information adopts user macro DO, output modal information and position information.

An interrupt program may execute in parallel with the original block. However, the machining will stop for a period of time at the end of the original block and before the end of the interrupt program.

A11.7. 4 Using the same program for general cutting and special cutting

(Specifications)

Each executive program is provided with special travel. But the instruction is not easy to be executed in general program.

(Parameter setting)

TSE=1: State triggering mode

MUSR=1: Enable user macro interrupt function

MSUB=*: Subprogram type/macro type user macro interrupt

MPRM=*: Setting of the M code for the control of user macro interrupt

MINT=1: Interrupt program is executed after the end of block.

(Explanations)

The interrupt program shall be instructed as follows

O××××
M97; ←——— Disable interrupt

⋮

M96; ←——— Enable interrupt

M99;

The interrupt signal is kept ON in state triggering mode.

Therefore, user macro interrupt is executed at the end of each block in the original program. The special travel to be executed is instructed in an interrupt program. The program part that needs no user macro interrupt is disabled using M97.

广州数控设备有限公司
GSK CNC EQUIPMENT CO., LTD.

Add: No.52, 1st . Street, Luochong North Road, Luochongwei, Guangzhou, 510165, China

Website: http://www.gsk.com.cn E-mail: gsk@gsk.com.cn
Tel: 86-20-81796410/81797922 Fax: 86-20-81993683

All specification and designs are subject to change without notice. Aug. 2007/Edition 4
Aug. 2007/Printing 4